NIA 95945-5002

THESIS
C 622

UTILIZATION OF A PC FOR FINITE ELEMENT MODELING

A Special Research Problem

Presented to

The Faculty of the School of Civil Engineering

by

Robin A. Collins

August, 1989

GEORGIA INSTITUTE OF TECHNOLOGY
A UNIT OF THE UNIVERSITY SYSTEM OF GEORGIA
SCHOOL OF CIVIL ENGINEERING
ATLANTA, GEORGIA 30332

T244059

UTILIZATION OF A PC FOR FINITE ELEMENT MODELING

A Special Research Problem

presented to

The Faculty of the School of Civil Engineering

Georgia Institute of Technology

by

Robin A. Collins

In Partial Fulfillment

of the Requirements for the Degree of

Master of Science in Civil Engineering

ACKNOWLEDGEMENTS

Thesis
C622
C.1

The author would like to express his gratitude to all those individuals and organizations who contributed to the completion of this special research project. First I would like to thank my wife, Brenda, and children, Michael and Patricia. To my wife who constantly provided the love, support and understanding required to go through with "this". To my son who continually provided me with the guidance to set my priorities straight and to my daughter who always met me at the door with a smile saying "glad your home". Additionally the author would like to thank his parents for instilling the perseverance required to complete this project once it was started.

Next I would like to thank the faculty and staff of the Geotechnical Engineering department of Civil Engineering. To Dr. Richard D. Barksdale, my advisor, for his guidance and constant concern on how things were going. Without his insight and assistance this project would not have been completed. To Dr. Robert C. Bachus, the most enthusiastic instructor I have ever had. His ability to assign "little" and "easy" homework assignments never ceases to amaze me and my fellow geotech students. To Ken Thomas for his companionship, humor, and "high" priority discussions on anything and everything. Finally to Vicki Clopton for her typing and other "key" support, humor and friendship.

Special gratitude is expressed to the United States Navy Civil Engineer Corps for providing the opportunity and financial support required to pursue a Master's Degree in Civil Engineering.

Next I would like to thank Mr. Ranga Chakravarty of Engineering Mechanics Research Corporation for his patience in answering my endless (if sometimes senseless) questions on the operation of their computer program.

Finally, I would like to thank all my fellow Geotech and nonGeotech students for their friendship and support. I would especially like to thank Jim Anderson, Mike Kemp, Bill Sheffield, and LCDR Chris Willis for their support and extra Tuesday and Thursday morning "tutorial" sessions.

TABLE OF CONTENTS

CHAPTER		PAGE
	Acknowledgements	i
1	Introduction	1.1
2	User's Manual	2.1
	Introduction	2.1
	Command Format	2.2
	Typical Commands	2.3
	General Executive Commands	2.8
	Command Rules	2.10
	Program Usage	2.11
	Helpful Hints	2.15
	Example Problems	2.17
	Static Example	2.19
	Command Input	2.21
	NISA II Input File	2.22
	Command Input Explanation	2.24
	Command Input Demonstration	2.39
	Eigenvalue Example	2.42
	Command Input	2.44
	NISA II Input File	2.45
	Command Input Explanation	2.51
	Command Input Demonstration	2.64
	Transient Dynamic Example	2.67
	Eigenvalue Input	2.69

CHAPTER PAGE

NISA II Eigenvalue Input File 2.70

Eigenvalue Input Explanation 2.71

Dynamic Input 2.84

Dynamic Input Explanation 2.85

Frequency Response Example 2.97

Eigenvalue Command Input 2.99

NISA II Eigenvalue Input File 2.100

Eigenvalue Input Explanation 2.101

Dynamic Input 2.103

Dynamic Input Explanation 2.114

3 Program Utilization and Comparison 3.1

Introduction 3.1

Frame with Slab on Springs 3.2

Circular Foundation on a Halfspace 3.12

4 Conclusions 4.1

5 Bibliography 5.1

Appendix A Required Equipment A.1

Appendix B Running Time B.1

Appendix C NISA II Executive Commands C.1

Appendix D NISA II vs GTSTRUDL Displacement Comparisons D.1

Appendix E Theoretical Displacement Calculations E.1

CHAPTER ONE

INTRODUCTION

INTRODUCTION

In the past, Geotechnical Engineering has always been sort of a "black science" as opposed to an "exact science", where the ability to predict what will occur beneath the earth's surface was often based primarily on experience. Over the years this has slowly changed. With the advent of the Finite Element method of analysis, Geotechnical Engineering leaped forward towards becoming a more exact science. The ability to predict is still based on experience. However the predictions are much better today.

Finite element analysis has a couple of draw backs though. One draw back is that mesh generation is very tedious and time consuming. The second draw back is that the overall mathematical analysis is also very time consuming. With the invention of the computer the mathematical analysis draw back is pretty much avoided, although at a substantial monetary cost. Performing a finite element analysis on a mainframe computer takes a lot of time and as a result cost a lot of money. As computers have become more sophisticated and quicker, the running time has decreased. However the cost per unit of time has increased. Therefore, a dynamic finite element analysis still costs a fair amount.

With the fast computers now available to perform finite element analyses, they became much more common. However they still have the first drawback of the tedious mesh generation. That is until programs were developed that perform mesh generation themselves.

Computers have continued to become more sophisticated. With the inception of the 386 generation personal computer (PC), many of them have a greater capability than the mainframe computers of yesteryear. In addition to their large capabilities these PCs are relatively inexpensive.

With the increasing capabilities of these PCs there there is an increasing amount of work that can be accomplished with them. As a result over the last forty years the engineering field has gone from the very tedious hand development of a finite element mesh and its' analysis, to just recently being able to use a PC to develop and analyze a finite element mesh.

This capability of using a PC to develop and analyze a finite element mesh has opened a whole new area of analyzing complex problems by the practitioner. Not only has it opened new areas but it has substantially decreased the cost. For the cost of performing three fairly complex analyses on a mainframe computer, a consulting firm can purchase a PC system capable of performing those same

analyses, a commercial finite element analysis program with mesh generation and post processing capability, and have personnel trained. As a result, the actual cost to the consultant for performing a fourth analysis is the personnel cost of developing the mesh, which would have occurred any way, and the minimal cost of computer and program maintenance.

There are many comprehensive finite element analysis programs, for dynamic and static problems, available on the market today. Programs such as STRUDL, NISA, ANSYS, and NASTRAN are readily available, to just name a few, and relatively easy to use. After obtaining an office computer system and finite element program package, a new user merely has to sit down and "use" the program to learn how it operates. This "use" should consist of first reading a few of the basic chapters in the user's manuals and then trying to develop and analyze problems with "known" solutions. Once the new user is able to obtain analysis responses that match the "known" solutions they are ready to go. As becoming intimately familiar with a comprehensive computer program may take many months, it is recommended that the new user "cross-check" the program output, with another method, until they are very comfortable with the program and its performance.

Until the start of this master's research problem the

author had very little computer experience and only a very basic understanding of what a finite element analysis was. This research problem consisted of providing a "new user" the opportunity of learning, from scratch, how to generate finite element meshes and analyze them using the NISA II computer program. This involved loading the program onto a PC, modeling problems that had solutions readily available (ie verification problems obtained with the user's manuals) and then analyzing problems and comparing the response to theoretical responses or those provided by a different finite element program (GTSTRUDL).

The following chapters will provide an individual that has no finite element experience and/or NISA II program experience with a more clarified user's manual, highlighting the problem areas the author encountered, and some comparisons between NISA answers and those provided by theory or another finite element program (GTSTRUDL).

Appendix A provides a list of the equipment required to operate the NISA II program and appendix B provides an idea of the time required to perform analyses of various sizes.

CHAPTER TWO

USER'S MANUAL

INTRODUCTION

There are basically three main methods of developing a finite element model for analysis. Method one consists of using a finite element modeling program like PATRAN. Once the model has been created NISA II can preform the analysis after the model has been interfaced with the DISPLAY-PRE program via a neutral file system. Method two consists of using a text or line editor, like EDLIN, and writing an ASCII file following the format in Sections 5 to 8 (input & executive commands, model data, analysis data, and modal dynamic data respectively) of the NISA II User's Manual. The third method consists of using the DISPLAY-PRE program portion of the NISA II package.

This user's manual will concentrate on how to develop finite element models using the DISPLAY-PRE program, and then how to perform a finite element analysis using the NISA II program. Since writing an all inclusive users manual is beyond the scope of this project, it is the author's intent that this users manual clarify and simplify the generation and analysis of finite element models using the NISA II documentation. Therefore, this chapter should be used concurrently with the actual NISA II User Manuals.

The DISPLAY-PRE program is a user friendly program that,

in addition to providing two modes for creating models (menu mode and command mode), automatically produces on the screen for viewing, a geometric representation of the model being created. The first mode of creating a model is the menu mode which guides the user through the various commands by way of a pyramid. This allows the user to select a "main" command and then allows the selection of an option from the "main" command's sub-menu. The second mode is the command mode. This allows the user to bypass the menu mode and input desired commands and sub-commands directly (for someone who is familiar with the program the command mode is the quicker method). The commands for both modes are identical. To access the command mode once the user is in the DISPLAY-PRE program merely type "c".

COMMAND FORMAT

The general format for the command input consists of:

Entry #	1	2	3	4	5
variable	name	function	input	output	data

Name: This is the "main" command.
Function: This is the desired option available from the "main" command.
Input: This is the list of identities that the function

uses to perform the command.

Output: This is the list of generated entities.

Data: This is the data used by the command to perform the required function.

TYPICAL COMMANDS

The following commands are available in the DISPLAY-PRE program and are typical of those used to create a finite element model for use in Geotechnical Engineering. For the complete list of all available commands refer to the DISPLAY II User's Manual.

Name	Description
GRD:	This stands for grid which is defined as a point in 3-D space. The user can create, add, delete, plot, search, etc. a grid using this command. Refer to pages 3.1-1 to 3.1-37 of the DISPLAY II User's Manual.
LIN:	This command is used to define a line. Lines can be created through two or more grids, plotted, deleted, etc. using this command. Refer to pages 3.2-1 to 3.2-64 of the DISPLAY-PRE User's Manual.

PAT: This command stands for patch and is used to create or manipulate a patch. A patch is a surface defined by parametric cubic equations. Refer to pages 3.3-1 to 3.3-46 of the DISPLAY II User's Manual. Refer to the eigenvalue example to see a method of creating a patch and what one looks like.

HYP: This command stands for hyperpatch and is similar to the PAT command only it is used to create a 3-D solid entity. Refer to pages 3.4-1 to 3.4-28 of DISPLAY II User's Manual.

FEG: This command stands for finite element generation and is used to create the finite element mesh. Refer to pages 4.1-1 to 4.1-28 of the DISPLAY II User's Manual.

FZM: This command stands for finite element generation with "topological zoom". With this command the user can create local high mesh densities without disturbing the rest of the mesh. Refer to pages 4.2-1 to 4.2-36 of the DISPLAY II User's Manual.

NOD: This command stands for node and can be used to

add, delete, search for, plot, etc. a node or list of nodes. Refer to pages 4.3-1 to 4.3-30 of the DISPLAY II User's Manual.

ELE: This command stands for element and is similar to the node command. Refer to pages 4.4-1 to 4.4-60 of the DISPLAY II User's Manual.

PRO: This command stands for property and is used to specify the geometric properties of a property index. For ease of providing the specified properties every element in the model is assigned a property index number. This allows the user to input one command to assign the geometric properties to a group of elements instead of inputing one command for each element. Refer to pages 4.5-1 to 4.5-8 of the DISPLAY II User's Manual.

MAT: This command stands for material and is used to specify the actual material properties (example Poisson's ratio) to a material index. Like the property index each element is assigned a material index. Therefore the user can assign a material property to a group of elements with one command. Refer to pages 4.6-1 to 4.6-8 of the DISPLAY II User's Manual.

The following example is provided to show how the property index and the material index is used to provide properties to various items. The item to be modeled consists of the following materials: a concrete beam 18 inches thick and E=4,074,280 psi (f'c=5000 psi), a concrete beam 18 inches thick and E=3,155,924 psi (f'c=3000 psi), a concrete slab (6 inches thick and E=4,074,280 psi (f'c=5000 psi), and a concrete slab 24 inches thick and E=3,155,924 psi (f'c=3000 psi). In this case the material and property index numbers could be assigned as follows:

Material	E (psi)	T (in)	MID #	PID #
beam	4,074,280	18	1	1
beam	3,155,924	18	2	1
slab	4,074,280	6	1	2
slab	3,155,924	24	2	3

FOR: This stands for force and allows the user to manipulate the forces in the model. Note: Forces are only applied to nodes. Refer to pages 4.7-1 to 4.7-14 of the DISPLAY II User's Manual.

PRE: This stands for pressure and allows the user to manipulate the pressures in the model. Note:

Pressures are only applied to elements. Refer to pages 4.8-1 to 4.8-10 of the DISPLAY II User's Manual.

DIS: This stands for displacement and allows the user to manipulate displacement constraints, ie boundary conditions, at specified nodes. Although the user's manual does not state it the author determined that the program will accept a maximum of ten variables between slashes "/" in entry 3 of this command. If greater than ten variables are required to define the displacement constraints, multiple DISP commands are required. Refer to the various verification examples and pages 4.9-1 to 4.9-14 of the DISPLAY II User's Manual.

GENERAL EXECUTIVE COMMANDS

DEC: This command places the program in the mode to accept Executive commands. Executive commands tell the program what type of analysis to perform, what files to save for post-processing, etc. Refer to page 5.1-1 of the NISA II User's Manual. Appendix C is a photocopy of table 5.2.3 on page 5.2-7 of the NISA II User's Manual and provides an alphabetical listing of all executive commands available.

INS: This is an executive command that allows the insertion of data lines.

ANAL=: This tells the program what type of analysis to perform. For example "ANAL=STATIC" tells the program to perform a static analysis.

FILE=: This tells the program the prefix to be used when saving the desired data files. For example "FILE=S01" tells the program that the prefix S01 is to be used when saving any binary data file. The author determined that using greater than five alphanumeric characters in this command will cause problems with the

program trying to retrieve the internally generated data files used for the dynamic analysis and post processing.
Note this command is to be used in conjunction with the following SAVE= command.

SAVE=: This tells the program what data files are to be saved. For example "SAVE=26,27" tells the program to save the internally generated data files 26 and 27. These files are named S0126.DAT and S0127.DAT respectively and are required if the user desires to perform any post-processing. Section 3 of the NISA II User's Manual, especially pages 3.15-1 and 3.15-2, describes a number of internally generated data files the program uses to perform its analysis. All of these files are temporary files and will not be saved unless, specifically saved with this command.

QUI: This command either takes the program out of the INS mode or the DEC mode depending on which mode the program is in when the QUI command is input.

COMMAND RULES

The general rules governing command input are:

Commands can either be typed in capital letters or lower case letters.

Commas "," separate the variables (ie name, function, input, output, data). Two consecutive commas indicate a null (void) entry. The program will automatically provide "DEFAULT" values to null entries, if applicable.

Slashes "/" separate multiple entries in one variable (example: name, function, input #1/input #2/input #3, etc.). Like commas, two consecutive slashes indicates a null (void) entry. Again the program will automatically provide "DEFAULT" values to null entries, if applicable.

Units: Any unit (example feet, foot-pound, meter, etc.) can be used in this program. The only requirement is that the user maintains unit consistency.

For a more in depth description of the general rules refer to page 2.1-3 of the DISPLAY II User's Manual.

MODEL GENERATION: The following steps are required to generate a finite element model:

Once the machine is on, get into the directory and sub-directory that the appropriate files will be stored in. For example on the machine the author used, all model generation was accomplished in the sub-directory "TEST" under the directory "EMRCNISA" on the C drive. Therefore, to create a model the author typed in "cd emrcnisa\test" once the C> was received. This changed the directory from the C directory to the sub-directory "TEST" of the directory "EMRCNISA".

Now type "NISA286" and hit the return. This will automatically access the main menu of the NISA II program, with the "DISPGDB" option highlighted. Hit return again to access the DISPLAY-PRE program.

The program is now ready to accept the commands required to generate the finite element model. As discussed above this can be accomplished either in the user friendly menu mode or the command mode.

To end the DISPLAY-PRE session type in the command "END".

RUN ANALYSIS: To perform an analysis the following steps are required:

Access the directory and sub-directory that the file

containing the model to be analyzed is in, then type "NISA286" and hit return.

Now move the cursor, using the arrow keys, to the desired analysis (Example: STATIC/EIGN/BUCK for either a static, eigenvalue or buckling analysis) mode and hit return. Note if a dynamic analysis is desired, an eigenvalue analysis must be performed first so that the natural frequencies of the model can be used in the actual dynamic analysis (for more info refer to the dynamic example).

No matter what type of analysis is to be performed the program will ask for the input file name at this time. For a static, eigenvalue, or buckling analysis type the file name generated with the "WRI,NISA,(file name).NIS" command and hit return. For a dynamic analysis type in the file name that was created using a text editor. Remember as stated above an eigenvalue analysis must be accomplished before a dynamic analysis can be completed.

Now the program will ask for the output file name. This is the file name the program will store the analysis in. For ease of file maintenance, it is recommended to use the same prefix as the input name and use the .out suffix (ie "file name.OUT"). Note: if an output file name is used that is currently employed in the originating sub-directory, the file in that sub-directory will be written over. After inputing the output file name hit the

return and wait for the analysis.

To end an analysis prior to its completion hit the control key "CTRL" and the "Break" key simultaneously. The program will then respond with "Strike any key when ready". At this time strike any key and the program will access the "main" menu. Now move the cursor, using the arrow keys, to "QUIT" and hit return. The program will now return back to the originating directory and sub-directory.

EXIT PROGRAM: To exit any portion of the DISPGDB (DISPLAY II) program, type end. The program will then ask if a data base file is to be written. Respond with either yes "Y" or no "N" and hit return. The program will now access the "main" menu. Now move the cursor, using the arrow keys, to "QUIT" and hit the return. The program will now return back to the originating sub-directory.

Refer to "RUN ANALYSIS" above for the method of exiting the program prior to the completion of an ongoing analysis.

VIEW A FILE: To view any file first access the directory and sub-directory that the file is in. Then use standard DOS commands such as "type" with the file name placed after the word "type" once a C>, D>, A>, etc. for the proper directory and sub-directory has been received. Example C>type s01.std; this will access the file s01.std

in the C drive and scroll it across the screen. To stop the screen from scrolling hit the control key, "CTRL" and the s key simultaneously. To start the scrolling again, hit any key.

PRINT A FILE: To print a file add a ">prn" after the above discussed type command. Example C>type s01.std>prn The fill s01.std in the C drive will now be printed.

HELPFUL HINTS

Based on the author's experience, because the NISA II and DISPLAY II programs are so user friendly, there really is a limited number of helpful hints that can be provided to the unfamiliar user. Most of the helpful hints are methods of avoiding problems when developing either the DISPLAY II file or the ASCII file, required for dynamic analysis. Wherever a problem or a limitation, not listed in the NISA II or DISPLAY II User's Manual, was discovered a method of avoiding this problem is provided. These methods are listed wherever they are appropriate, whether it be in the general command, Executive command, or verification problem section of chapter two. Additionally, the following hints are provided:

1. Whether the user is experienced or a novice it is recommended that the user sit down and develop the model, to be analyzed, with paper and pencil prior to sitting at the key board. The main reason is, it is much easier to erase an unwanted or incorrect data line on paper than it is to delete one once it has been inputed into the DISPLAY II program. In fact, some command lines can not be deleted. As a result if one of these lines is incorrectly inputed, the user must start completely over inputing the required model generation commands to correct the model.

2. Keep the hard disk free of unwanted clutter and unnecessary files. Due to the program storing numerous

temporary files when in the process of performing an analysis a 10,000 degree of freedom model requires approximately 12 megabytes of disk space. A 10,000 DOF system may seem excessive for a Geotechnical Engineer and therefore why worry about space? The reason this "house keeping" needs to be accomplished is because a 1350 DOF system, which is easily within the range a Geotechnical Engineer would use, requires approximately 2 megabytes of disk space. If there is not enough disk space available two things will happen: 1. the program will not complete the analysis and 2. because the program does not complete its' analysis all of the temporary files created remain on the disk until deleted. The author at one point attempted a 1350 DOF model with 1.8 megabytes of space available and ended up with the program "crashing", 0 bytes of disk space available, and approximately 20 temporary files on the disk that had to be deleted. When the system "crashed" these temporary files are stored with the prefix the user inputed with the Executive command "File=" and the suffix ".tmp". For example if the Executive command "File=s01" was used and the system "crashed" one of the temporary files saved on the disk may be "s0128.tmp". Note: all the internally generated files are numbered 26., 27., 28., 32., etc.

EXAMPLE PROBLEMS

Because the NISA II Verification Problems Manual was written prior to the development of the DISPLAY-PRE program, those problems do not provide the DISPLAY-PRE commands required for model generation. As a result the following problems were selected from the Verification Problems Manual, as good representative examples of what a Geotechnical Engineer may encounter, for a step by step description of how to develop the model using DISPLAY-PRE and then how to perform the desired analysis.

In order to complete a dynamic analysis of a structure, facility, mass etc. two files are required to "design" the finite element model. The first file contains the actual geometric and material properties of the model and is developed using the DISPLAY II program. This file is then used to complete an eigenvalue analysis of the model. The eigenvalue analysis determines the natural frequencies of the system, which is then used in the dynamic analysis. The second file is written using a standard text or line editor, like EDLIN, and contains the loading characteristics of the model. Additionally, this file contains the commands required to tell the program what the desired output is. This file is generated following Section 8 "MODAL DYNAMIC DATA" of the NISA II User's Manual.

The following examples consists of the input file

(the actual commands inputed during the DISPLAY-PRE session), the NISA file (the actual file that is created by the DISPLAY-PRE pre-processing program and is used during the finite element analysis), and a line by line description of the input file. Additionally, the dynamic problems contain the dynamic input file created using the text or line editor and a line by line description of this input file.

The line by line descriptions discussed above do not discuss every line of the input file. However, they do discuss every different line of input. If there is more than one line of input performing the same function, for example a model may contain five lines of input listing the movement constraints the model will have, only one of the input lines is discussed.

VERIFICATION PROBLEM 2.20

TITLE:

Beam on a tensionless elastic foundation

ELEMENT TYPE:

2-D beam element (NKTP = 13, NORDR = 1)
2-D spring element (NKTP = 18, NORDR = 1)
2-D gap element (NKTP = 42)

PROBLEM:

A 30 inch long beam (3 inches wide, 1 inch deep) rests on an elastic soil foundation (Fig. 2.20.1). The soil is active in compression only. A concentrated load of 8000 lbs. is applied at the midspan of the beam. The foundation modulus is 5000 lb/in^3.

The midspan displacement and the contact length of the beam are calculated.

PROPERTIES:

Material:

EX	=	30.0E6 psi	(Modulus of elasticity of beam material)
NUXY	=	0.0	(Poisson's ratio)

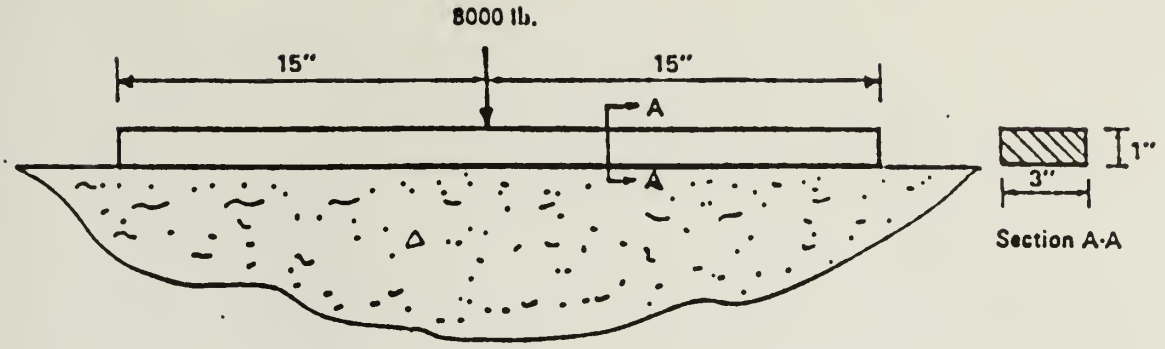
Cross-section:

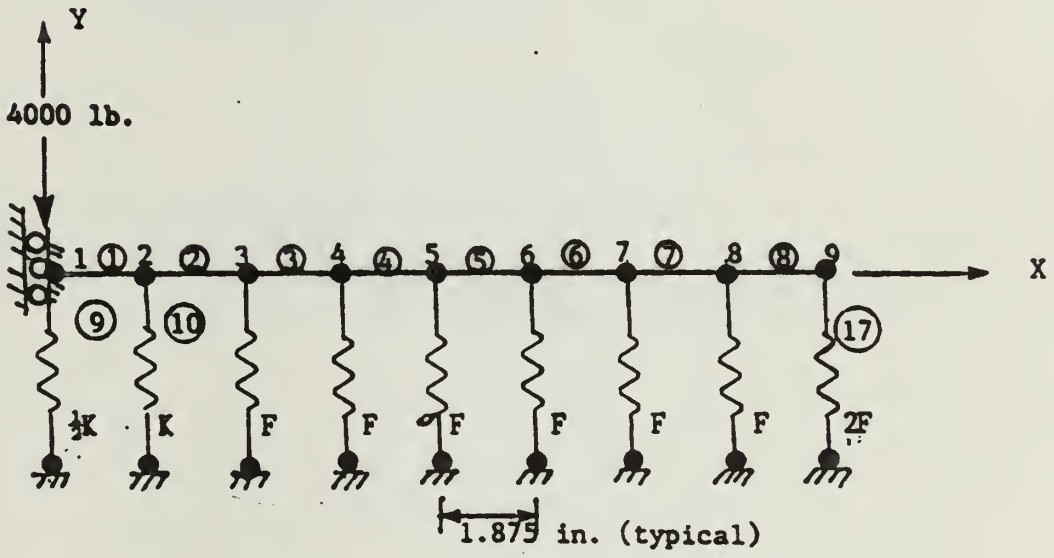
IZZ	=	0.25 in^4	(Second area moment about beam Z-axis)
A	=	3 in^2	(Cross-sectional area)

FINITE ELEMENT MODEL:

Due to symmetry, only one half of the structure is modeled using 2-D beam, 2-D spring and 2-D gap (or compression-only spring) elements (Fig. 2.20.1). The beam is divided into eight equal length beam elements with two nodes per element. The tensionless foundation is simulated by nine discrete equidistant springs. The two springs (near the midspan) are assumed to be active in tension and compression. These two springs support the beam initially. The remaining seven compression-only springs are modeled by gap elements with negative flexibility coefficients and zero gap width.

The spring constant (K) is taken as foundation modulus times beam-width times spacing between the springs (K = 5000*3*1.875 = 28125 lb/in). The flexibility coefficient (F) for a typical compression-only spring is the reciprocal of the spring constant (F = -1/K).





K ; spring constant = -1/F,

F : flexibility coefficient for a compression-only member

Figure 2.201 Geometry, loading and finite element model for a beam on elastic foundation.

```
GRD,ADD,1,0/0/0
GRD,TRS,1,2,15
LIN,2GD,1,2,1
FEG,BAR,1,,,8/13,1/1
LIN,TRS,1,2,/-10
FEG,BAR,2,,,8/13
ELE,DEL,9T16
ELE,ADD,9,18/1//2,1/10
ELE,ADD,10,18/1//3,2/11
ELE,ADD,11,42/1//4,3/12
ELE,ADD,12,42/1//4,4/13
ELE,ADD,13,42/1//4,5/14
ELE,ADD,14,42/1//4,6/15
ELE,ADD,15,42/1//4,7/16
ELE,ADD,16,42/1//4,8/17
ELE,ADD,17,42/1//5,9/18
DISP,ADD,1,1,0/////0
DISP,ADD,2T9,1,0
DISP,ADD,10T18,1,0/0
FORCE,ADD,1,1,/-4000.
PROP,ADD,1,3/0.25/0/0
MAT,ADD,1,30.0E6/0.0
PROP,ADD,2,14062.5
PROP,ADD,3,28125
PROP,ADD,4,0./-0.35555E-4/0./1.0E-8/-90
PROP,ADD,5,0./-0.71111E-4/0./1.0E-8/-90
DEC
INS
ANAL=STATIC
FILE=S01
SAVE=26,27
*TITLE,ATTEMPT OF VERIFICATION PROBLEM S20.STD
*I3,1,0,1,0,0,0,0,0,0,0.0000E-01,0.0000E-01
*I4,1,POINT LOAD AT BEAM'S CENTER
QUI
QUI
WRI,NISA,S01.NIS
WRI,DBS,S01.BIN
END
N
```

```
** EXECUTIVE
ANAL=STATIC
FILE=S01
SAVE=26,27
*TITLE
ATTEMPT OF VERIFICATION PROBLEM S20.STD
*ELTYPE
  1,  13,      1
  2,  18,      1
  3,  42,      1
*RCTABLE
       1,      4
   3.0000,    0.2500,    0.0000,    0.0000,
       2,      1
14062.5
       3.      1
28125
       4,      5
0.0000,-.35555E-4,0.,1.E-8,-90
       5.      5
0.0000,-.71111E-4,0.,1.E-8,-90
*ELEMENT
       1,      1,      1,      1, 0
       1,      2,
       2,      1,      1,      1, 0
       2,      3,
       3,      1,      1,      1, 0
       3,      4,
       4,      1,      1,      1, 0
       4,      5,
       5,      1,      1,      1, 0
       5,      6,
       6,      1,      1,      1, 0
       6,      7,
       7,      1,      1,      1, 0
       7,      8,
       8,      1,      1,      1, 0
       8,      9,
       9,      0,      2,      2, 0
       1,     10,
      10,      0,      2,      3, 0
       2,     11,
      11,      0,      3,      4, 0
       3,     12,
      12,      0,      3,      4, 0
       4,     13,
      13,      0,      3,      4, 0
       5,     14,
      14,      0,      3,      4, 0
       6,     15,
      15,      0,      3,      4, 0
       7,     16,
      16,      0,      3,      4, 0
       8,     17,
      17,      0,      3,      5, 0
       9,     18,
```

```
       1 ,,,,       0.0000,      0.0000,      0.0000,
       2 ,,,,       1.8750,      0.0000,      0.0000,
       3 ,,,,       3.7500,      0.0000,      0.0000,
       4 ,,,,       5.6250,      0.0000,      0.0000,
       5 ,,,,       7.5000,      0.0000,      0.0000,
       6 ,,,,       9.3750,      0.0000,      0.0000,
       7 ,,,,      11.2500,      0.0000,      0.0000,
       8 ,,,,      13.1250,      0.0000,      0.0000,
       9 ,,,,      15.0000,      0.0000,      0.0000,
      10 ,,,,       0.0000,    -10.0000,      0.0000,
      11 ,,,,       1.8750,    -10.0000,      0.0000,
      12 ,,,,       3.7500,    -10.0000,      0.0000,
      13 ,,,,       5.6250,    -10.0000,      0.0000,
      14 ,,,,       7.5000,    -10.0000,      0.0000,
      15 ,,,,       9.3750,    -10.0000,      0.0000,
      16 ,,,,      11.2500,    -10.0000,      0.0000,
      17 ,,,,      13.1250,    -10.0000,      0.0000,
      18 ,,,,      15.0000,    -10.0000,      0.0000,
*MATERIAL
 EX    ,       1 ,  ,   0.30000E+08
 NUXY  ,       1 ,  ,   0.00000E+00
*LDCASE
0,1,0,0,0,0,0,0.0000E-01,0.0000E-01
*LCTITLE
POINT LOAD AT BEAM'S CENTER
*SPDISP
** SET ID =        1
       1,ROTZ,     0.000000
       1,UX   ,    0.000000
       2,UX   ,    0.000000
       3,UX   ,    0.000000
       4,UX   ,    0.000000
       5,UX   ,    0.000000
       6,UX   ,    0.000000
       7,UX   ,    0.000000
       8,UX   ,    0.000000
       9,UX   ,    0.000000
      10,UY   ,    0.000000
      10,UX   ,    0.000000
      11,UY   ,    0.000000
      11,UX   ,    0.000000
      12,UY   ,    0.000000
      12,UX   ,    0.000000
      13,UY   ,    0.000000
      13,UX   ,    0.000000
      14,UY   ,    0.000000
      14,UX   ,    0.000000
      15,UY   ,    0.000000
      15,UX   ,    0.000000
      16,UY   ,    0.000000
      16,UX   ,    0.000000
      17,UY   ,    0.000000
      17,UX   ,    0.000000
      18,UY   ,    0.000000
      18,UX   ,    0.000000
 *CFORCE
 ** SET ID =        1
       1,FY   ,-0.40000E+04
 *ENDDATA
```

```
COMMAND>GRD,ADD,1,0/0/0
```

This adds a GRID (a point in 3-D space), calls it grid #1, and puts it at coordinates (0,0,0).

```
COMMAND>GRD,TRS,1,2,15
```

This makes grid #2 by "translating" grid #1 15 units in the X direction (actually it translates grid #1 15/0/0 units where the format is X/Y/Z; since there are no "0"s typed in, they are automatically assigned). If it was desired that grid #2 be placed 15 units in the Y direction, the input would be either "GRD,TRS,1,2,0/15" OR "GRD,TRS,1,2,/15". Default values are "0".

```
COMMAND>LIN,2GD,1,2,1
```

This creates a line starting at grid #1 and ending at grid #2, calling it line #1. The format is "LIN,2GD,starting grid #,ending grid #, line #".

```
COMMAND>FEG,BAR,1,,,8/13,1/1
```

FEG stands for "Finite Element Generation". This entry creates a group of 8 bar/beam elements (of element type 13), in the global coordinate system, assigning a

material ID # of 1 and a property ID # of 1, and a zoom of 1. Each element is defined with a node on each end and both nodes and elements are automatically numbered by the program. This is accomplished using the following format: "FEG,BAR,3,4,5,6A/6B/6C,7A/7B/7C" where:

Entry 3 is the number of lines to be "FEGED"

Entry 4 is the list of generated elements. If left blank the program will automatically number the elements.

Entry 5 is the list of generated nodes. If left blank the program will automatically number the nodes.

Entry 6A is the number of elements to be generated for each line.

Entry 6B is the NKTP #, which stands for the element type identification number. Each element type has a different NKTP #. For an explanation of exactly what a specific NKTP # is, refer to the element library section of the NISA II Users Manual, pages 4.0-1 to 4.120-5.

Entry 6C is the node # that has a local coordinate system the user may wish to use. This entry is used only if the beam is not to be defined in the global coordinate system (ie the Cartesian, Cylindrical, or Spherical systems). At times it is more convenient to refer to items with a local coordinate system,

for example a force or displacement constraint may be defined with directions which are not aligned with the global coordinate system. For more information on local coordinate systems refer to pages 3.2-1 to 3.2-4 of the NISA II User's Manual.

Entry 7A & 7B are the material and property ID #'s respectfully. This set of #s, assigned to one or more elements, is used later in the input phase for assigning actual properties. If left blank the program automatically assigns a material ID # of 1 and a property ID # of 1. These numbers do not have to be the same. A group of elements can have the same material properties (Poisson's ratio, Young's modulus, etc) with some of the elements having one geometric property and others having a different geometric property. As a result this group of elements would have one material ID # and two property ID #s.

Entry 7C is the zoom factor, ie the length of the last element to the length of the first element. If all elements are the same length then the zoom=1. If left blank the program automatically assigns a value of 1. For more information and a pictorial representation refer to page B-3 of the DISPLAY II User's

Manual.

```
COMMAND>LIN,TRS,1,2,/-10
```

This creates a second line, #2, by "translating" line #1 -10 units in the Y direction.

```
COMMAND>FEG,BAR,2,,,8/13
```

This creates a group of 8 bar/beam elements (of NKTP type 13) along line #2, each element is defined with a node on each end, and each node and elements is automatically numbered by the program. This was done solely to create the nodes required for defining the spring elements. Note, the elements of this line are not needed and will be deleted later. Another method of generating these nodes would be by generating each individual node using the command "NOD,ADD,NODE #,X/Y/Z (COORDINATES)/LOCAL COORDINATE SYSTEM #". By using the FEG command the nine required nodes, for the required springs, can be created at the same time. By using the NOD command creating the nine nodes would require multiple NOD commands.

```
COMMAND>ELE,DEL,9T16
```

This deletes elements 9 thru 16. Even though the

elements are deleted the nodes still remain.

COMMAND>ELE,ADD,9,18/1//2,1/10

This adds an element #9 with NKTP=18, order=1, material ID # =0, property ID # =2 between nodes 1 and 10, using the following format:
"ELE,ADD,3,4A/4B/4C/4D,5A/5B,6" where:

Entry 3 is the element # to be added. If this is left blank the program will automatically assign a number.

Entry 4A is the NKTP # (see entry 6B under the FEG description above).

Entry 4B is the order #. Some elements only have one order, NORDR=1, others have many depending on the element type. For a pictorial description of how this order works/varies refer to section 4, the "element library" of the NISA II User's Manual; specifically NKTP #s 1,2,3,4,5,& 32 provide good examples of some of the element types with varying orders.

Entry 4C is the material ID #. Since these are spring elements the material characteristics are included with the property characteristics (see the description of NKTP # 18 in the NISA II User's Manual). Therefore, this element does not need a material ID number. Accordingly it

is "0".

Entry 4D is the property ID #. (See entry 7B under the FEG description above.)

Entry 5A & 5B are the two nodes that defines the element.

Entry 6 is the vector number of a previously defined vector used in establishing a local coordinate system. This can be used for elements of NKTP = 11,12,& 38.

```
COMMAND>DISP,ADD,1,1,0/////0
```

This constrains node 1 against movement in the X direction and against rotation in the Z direction, and assigns this a constraint ID # of 1 with the following format: "DISP,ADD,3,4,5A-5F", where:

Entry 3 is the list of nodes with the assigned constraint. If it was desired that all nodes from 1 to 10 be constrained, with the same constraint, the input would be "1T10".

Entry 4 is the constraint ID #. The set ID under which the constraints are specified.

Entry 5A to 5F are the degree(s) of freedom that are constrained. This is accomplished by placing a "0" in the degree of freedom that must be constrained with the following format: "X/Y/Z/ROTX/ROTY/ROTZ". If the desired

constraints are in the Y and ROTX directions the input would be either "/0//0" or "/0//0//". Consecutive slashes indicate a null entry which the program identifies and either ignores or places a default value in the location. In this case the program ignores the entry. For more information on null entries refer to the general command rules previously addressed in this chapter or to pages 2.1-3 to 2.1-6 of the DISPLAY II User's Manual.

COMMAND>FORCE,ADD,1,1,/-4000

This entry adds a force of -4000 pounds in the Y direction, at node 1, with the following format: "FOR,ADD,3,4,5", where:

Entry 3 is the node that the force is applied to.

Entry 4 is the force set #. A value greater than 1 is used for multiply loading conditions. Example: suppose the response of a model to different loading conditions is desired, like different load amounts at the same node and/or the same or different forces at different nodes. To do this one force condition would be identified with one force set # and the other force condition would have a different force set

#. This allows a model to be analyzed with different loading conditions at the same time instead of performing one analysis with one loading condition and then remodeling the model with the different loading condition. In this case suppose it is desired that the systems response be determined for a force of -4000 lbs and 7500 lbs, both in the Y direction. The input would be: COMMAND>"FORCE,ADD,1,1,/-4000" AND COMMAND>"FORCE,ADD,1,2,/7500".
When the analysis is performed, answers will be received for both loading conditions.
Entry 5 is the force value in the desired direction X/Y/Z/MX/MY/MZ. Example: if the desired load condition is a force of 100 lbs in the Z direction and a moment of 10 ft.lbs. in the Y direction the input would be either "//100//10" or "0/0/100/0/10/0". If a position is left blank the program will automatically place a "0" in it.

COMMAND>PROP,ADD,1,3/0.25/0/0

This adds a cross sectional area of 3, a moment of inertia of 0.25, a transverse shear coefficient of 0, and a local y-coordinate for stress recovery of 0 to property ID set #1. A local y-coordinate of "0" means that there

is no local coordinate system, rather the global coordinate system is being used. This command has the following format "PROP,ADD,3A/3B/3C...,4", where:

Entry 3 is the property ID set that the properties will be assigned to.

Entry 4 is the list of properties which are defined by the element type with each different property being separated by a /. (Refer to the specific element type in the element library section of the NISA II Users Manual for more info on what properties are listed here).

Entry 5 is the section orientation for the beam element.

Note: there seems to be a couple of problems with this command. First the input field is a maximum of 10 characters, and the program automatically places up to four "0" after the decimal point. Therefore, if the input is a five figure number (ex 28125) the program will list ********** in the (file name).NIS file. Second the program does not seem to accept any numbers in the second or fourth fields of entry number 3, automatically assigning its own numbers. Therefore, these lines must be manually changed in the (file name).NIS file using a text or line editor like EDLIN. The (filename).NIS is discussed in more detail later.

COMMAND>MAT,ADD,1,30.0E6/0.0

This adds the material properties of Youngs modulus of elasticity = 30.0E6 psi and Poissons ratio = 0.0 to material ID set #1 with the following format "MAT,ADD,3,4A/4B/4C/..." where:

Entry 3 is the material ID set #.

Entry 4 is the list of material properties. Like the property characteristics described above, this list will vary for each element type.

COMMAND>DEC

This places the program in the mode to accept Executive Commands. For more info on executive commands refer to the previous Executive command overview and pg 7.10-1 of the DISPLAY II User's Manual.

COMMAND>INS

This places the program in the mode to allow the insertion of data lines.

COMMAND>ANAL=STATIC

This tells the program that the analysis to be performed is a STATIC analysis.

COMMAND>FILE=S01

This provides the file name prefix S01 for the internally generated data files.

COMMAND>SAVE=26,27

This tells the program to save files 26 and 27, internally generated and named files. The name of these files will be S0126.DAT and S0127.DAT respectively. Note if post processing is to be performed files 26.dat and 27.dat must be saved, which is accomplished using this command. Refer to the previous Executive command overview and section 3 of the NISA II User's Manual.

COMMAND>*TITLE,ATTEMPT OF VERIFICATION PROBLEM S20.STD

This tells the program that the title of this analysis is "ATTEMPT OF VERIFICATION PROBLEM S20.STD", which the program will then print at the top of the answer output.

COMMAND>*I3,1,0,1,0,0,0,0,0,0.0000E-01,0.0000E-01

The *I3 is the same as *LDCASE which is the load case "control card" for the desired output. Refer to page 7.1-2 of the NISA II Users Manual for an explanation of what each number means. Note: there is a problem with this

command on the PC version of 88.7. No matter what entry is placed after "*I3," the program automatically changes it to "0,1,3,0,0,4,0" on the *LDCASE card in the "(file name).NIS" file. The "(file name).NIS file is the file that the NISA II program actually uses for the analysis. Therefore, if different output than what is specified with "0,1,3,0,0,4,0" is desired, the *LDCASE line in the (file name).NIS file must be manually changed. This can be accomplished using a text or line editor like EDLIN.

COMMAND>*I4,1,POINT LOAD AT BEAM'S CENTER

The *I4 is the same as *LCTITLE which means load case title. This is the title for the specific load case. In this example the title "POINT LOAD AT BEAM'S CENTER" is assigned to load case 1. If there were multiple load cases, as discussed in "FORCE,ADD,..." above, multiple load case title cards can be used, one for each load case. In the description for entry 5 of "FORCE,ADD..." two loadings were discussed, one force and one moment. In that case the input could be:

COMMAND>*I4,1,POINT LOAD

COMMAND>*I4,2,MOMENT

or what ever the desired titles were. The program will then place the specified title at the heading of the answers for each load case.

COMMAND>QUI

Quits the insert mode.

COMMAND>QUI

Quits the DEC mode.

COMMAND>WRI,NISA,S01.NIS

This takes the above data just input during this session and writes a "NISA" file called "S01.NIS" that the program uses for it's analysis.

COMMAND>WRI,DBS,S01.BIN

This takes the above data and writes a binary data base file called "S01.BIN".

COMMAND>END

This ends the session.

Data Base File: At this time the program will ask if a data base file DISP.BIN is to be written. Since this has already been done with the WRI,DBS,S01.BIN command above, answer no "N".

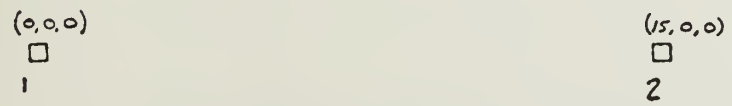
The model is now ready to be analyzed. Refer to the previous section "Program Usage" for guidance on how to perform the analysis.

COMMAND INPUT DEMONSTRATION

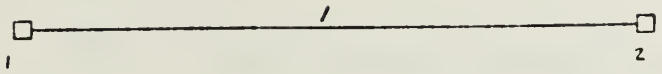
In an attempt to assist the unfamiliar user with understanding what the various DISPLAY-PRE commands actually do, the demonstration below provides a pictorial description of some of the various commands required in the previous verification problem. This demonstration does not contain all required commands only a few to provide an idea of how the commands work.

GRD,ADD,1,0/0/0 This creates a point at coordinates (0,0,0).

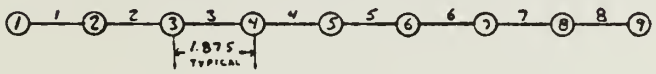
GRD,TRS,1,2,15 This creates grid #2 by translating grid #1 15 units in the X direction.



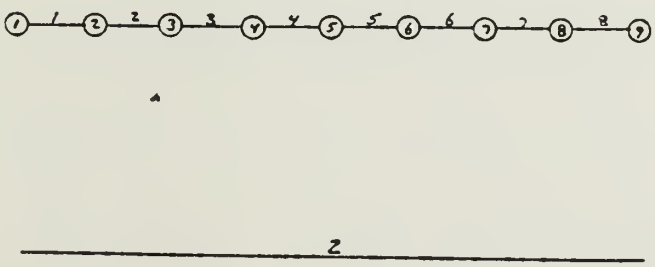
LIN,2GD,1,2,1 This creates line #1 between grids #1 and #2.



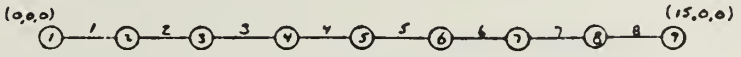
FEG,BAR,1,,,8/13,1/1 This creates 8 elements of NKTP type #13 along line #1, and assigns MID=1 and PID=1. Note there is a node created at the end of each element.

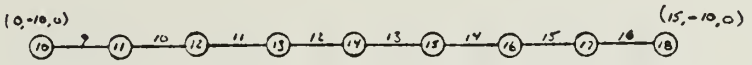


LIN,TRS,1,2,/-10 This creates line #2 by translating line #1 -10 units in the Y direction.

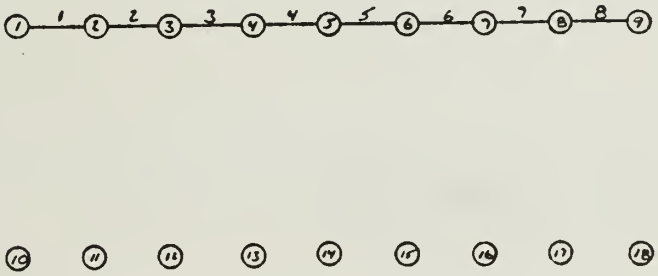


FEG,BAR,2,,,8/13 This creates 8 elements of NKTP type 13 along line #2 and allows the program to automatically number them. Again, as mentiond above, each element has a node at each end.

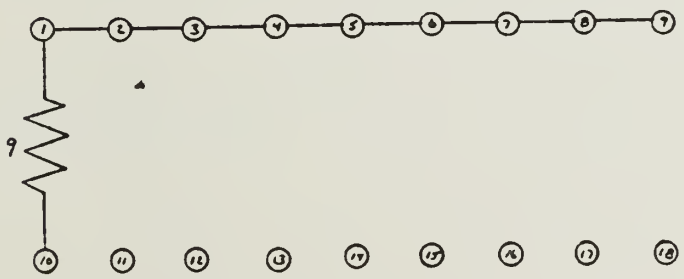




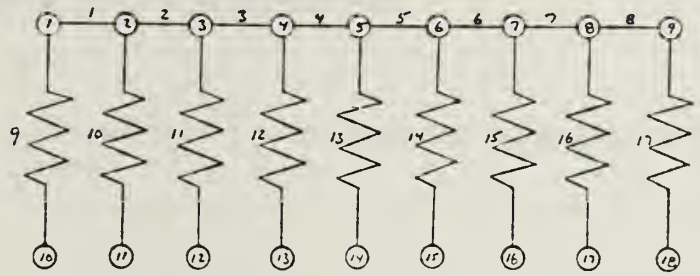
ELE,DEL,9T16 This deletes elements 9 through 16. However, nodes 10 to 18 remain.



ELE,ADD,9,18/1//2,1/10 This adds element 9, of NKTP type 18 and PID #2, between nodes 1 and 10.



ELE,ADD,17,42/1//5,9/18 This adds element 17, of NKTP type 42 and PID #5 between nodes 9 and 18. Note the steps to add elements 10 to 16 were not shown, but would consists of basically the same command.



VERIFICATION PROBLEM 5.5

TITLE:

Natural frequencies of a simply supported square plate

ELEMENT TYPE:

3-D general shell element (NKTP = 20, NORDR = 2)

PROBLEM:

The first five natural frequencies of a simply supported square plate, with dimensions as shown in Fig. 5.5.1, are to be computed.

PROPERTIES:

Material:

EX	=	10.5E6 psi	(Modulus of elasticity)
NUXY	=	0.3333	(Poisson's ratio)
DENS	=	2.588E-4 lbs.-sec^2/in.^{4}	(Mass density)

Geometry:

t = 1.0 in (thickness)

FINITE ELEMENT MODEL:

The plate is modeled using sixteen 3-D general shell elements (NKTP = 20, NORDR = 2). The nodes on the periphery of the plate are constrained in the normal (UZ) direction. In-plane translations (UX and UY) for all the nodes are constrained to obtain only the bending modes.

SOLUTION PROCEDURE:

Conventional subspace iteration and lumped mass matrix formulation are used to extract the eigenvalues.

RESULTS & COMPARISON:

The first five natural frequencies of the simply supported plate obtained from NISA are compared to the theoretical (Ref. [1]) values in Table 5.5.1. The mode shape plots are shown in Figs. 5.5.2 through 5.5.6.

REFERENCE:

1. S. Timoshenko, D. H. Young, and W. Weaver, Jr., "*Vibration Problems in Engineering*", 4th edition, John Wiley & Sons, New York (1974).

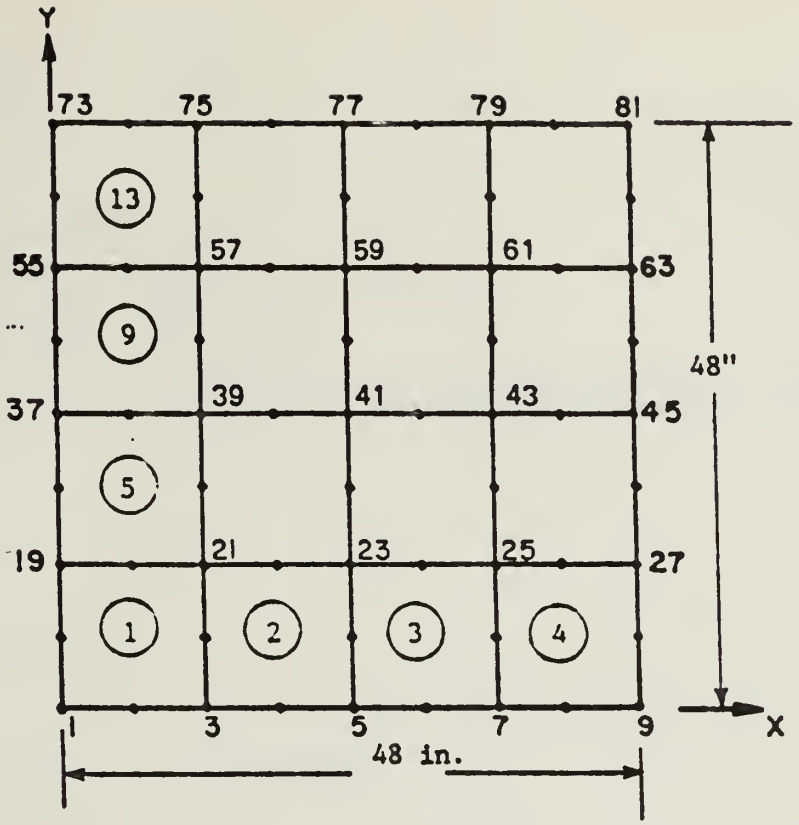


Figure 5.5.1 Finite Element Mesh for a Simply Supported Square Plate

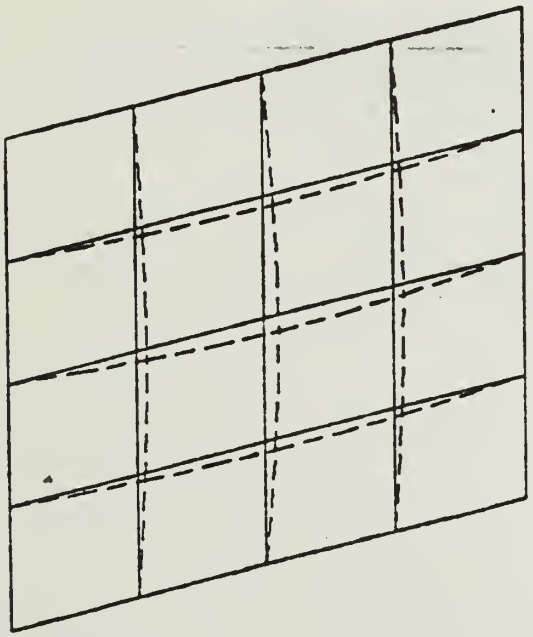


Figure 5.5.2 Mode No. 1: Frequency = 83.979 Hz

```
GRD,ADD,1,0/0/0
GRD,TRS,1,2,48
LIN,2GD,1,2,1
LIN,TRS,1,2,0/48
PAT,2LN,1,2,1
FEG,QUA,1,,,2/4/4/20
PROP,ADD,1,1/1/1/1/1/1/1/1
MAT,ADD,1,10.5E6/0.3333/////////2.5879E-4
DISP,ADD,1/9/57/65,1,0/0/0/0/0
DISP,ADD,2T8/58T64,1,0/0/0//0
DISP,ADD,10/14T15/23T24/28T29/37T38/42T43/51T52/56,1,0/0/0/0
DISP,ADD,11T13/16T22/25T27/30T36/39T41/44T50/53T55,1,0/0
DEC
INS
ANAL=EIGEN
FILE=E01
SAVE=26,27
RESE=ON
AUTO=ON
EIGE=SUBS,CONV
MASS=LUMPED
*LDCASE,1
*EIGCNTL,5,0,20,0,0.0,0.0,1.0E-5
*MODEO,1,1,1,1
*EIGO,1,1,0,3,1,0,1
QUI
QUI
WRI,NISA,E01.NIS
WRI,DBS,E01.BIN
END
N
```

```
** EXECUTIVE
ANAL=EIGEN
FILE=E01
SAVE 26,27
RESE ON
AUTO-ON
EIGE=SUBS,CONV
MASS=LUMPED
*ELTYPE
  1,  20,     2
*RCTABLE
      1,     8
   1.0000,   1.0000,   1.0000.   1.0000,   1.0000,   1.0000,   1.0000,   1.00
*ELEMENT
      1,     1,     1,     1, 0
      1,     2,     3,    11,    17,    16.    15,    10,
      2.     1,     1,     1, 0
      3,     4,     5,    12,    19,    18,    17,    11,
      3,     1,     1.     1, 0
      5,     6,     7,    13,    21.    20.    19.    12,
      4,     1,     1,     1, 0
      7,     8,     9,    14,    23,    22,    21,    13,
      5,     1,     1,     1, 0
     15,    16,    17,    25,    31,    30,    29,    24,
      6,     1,     1,     1, 0
     17,    18,    19,    26,    33,    32,    31,    25,
      7,     1,     1,     1, 0
     19,    20,    21,    27,    35,    34,    33,    26,
      8,     1,     1,     1, 0
     21,    22,    23,    28,    37,    36,    35,    27,
      9,     1,     1,     1, 0
     29,    30,    31,    39,    45,    44,    43,    38,
     10,     1,     1,     1, 0
     31,    32,    33,    40,    47,    46,    45,    39,
     11,     1,     1,     1, 0
     33,    34,    35,    41,    49,    48,    47,    40,
     12,     1,     1,     1, 0
     35,    36,    37,    42,    51,    50,    49,    41,
     13,     1,     1,     1, 0
     43,    44,    45,    53,    59,    58,    57,    52,
     14,     1,     1,     1, 0
     45,    46,    47,    54,    61,    60,    59,    53,
     15,     1,     1,     1, 0
     47,    48,    49,    55,    63,    62,    61,    54,
     16,     1,     1,     1, 0
     49,    50,    51,    56,    65,    64,    63,    55,
*NODES
      1 ,,,,      0.0000,     0.0000,     0.0000,
      2 ,,,,      6.0000,     0.0000,     0.0000,
      3 ,,,,     12.0000,     0.0000,     0.0000,
      4 ,,,,     18.0000,     0.0000,     0.0000,
      5 ,,,,     24.0000,     0.0000,     0.0000,
      6 ,,,,     30.0000,     0.0000,     0.0000,
      7 ,,,,     36.0000,     0.0000,     0.0000,
      8 ,,,,     42.0000,     0.0000,     0.0000,
      9 ,,,,     48.0000,     0.0000,     0.0000,
```

```
10 ,,,,     0.0000,     6.0000,     0.0000,
11 ,,,,    12.0000,     6.0000,     0.0000,
12 ,,,,    24.0000,     6.0000,     0.0000,
13 ,,,,    36.0000,     6.0000,     0.0000,
14 ,,,,    48.0000,     6.0000,     0.0000,
15 ,,,,     0.0000,    12.0000,     0.0000,
16 ,,,,     6.0000,    12.0000,     0.0000,
17 ,,,,    12.0000,    12.0000,     0.0000,
18 ,,,,    18.0000,    12.0000,     0.0000,
19 ,,,,    24.0000,    12.0000,     0.0000,
20 ,,,,    30.0000,    12.0000,     0.0000,
21 ,,,,    36.0000,    12.0000,     0.0000,
22 ,,,,    42.0000,    12.0000,     0.0000,
23 ,,,,    48.0000,    12.0000,     0.0000,
24 ,,,,     0.0000.    18.0000,     0.0000,
25 ,,,,    12.0000,    18.0000,     0.0000,
26 ,,,,    24.0000,    18.0000,     0.0000,
27 ,,,,    36.0000,    18.0000,     0.0000,
28 ,,,,    48.0000,    18.0000,     0.0000,
29 ,,,,     0.0000,    24.0000,     0.0000,
30 ,,,,     6.0000,    24.0000,     0.0000,
31 ,,,,    12.0000,    24.0000,     0.0000,
32 ,,,,    18.0000,    24.0000,     0.0000,
33 ,,,,    24.0000,    24.0000,     0.0000,
34 ,,,,    30.0000,    24.0000,     0.0000,
35 ,,,,    36.0000,    24.0000,     0.0000,
36 ,,,,    42.0000,    24.0000,     0.0000,
37 ,,,,    48.0000,    24.0000,     0.0000,
38 ,,,,     0.0000,    30.0000,     0.0000,
39 ,,,,    12.0000,    30.0000,     0.0000,
40 ,,,,    24.0000,    30.0000,     0.0000,
41 ,,,,    36.0000,    30.0000,     0.0000,
42 ,,,,    48.0000,    30.0000,     0.0000,
43 ,,,,     0.0000,    36.0000,     0.0000,
44 ,,,,     6.0000,    36.0000,     0.0000,
45 ,,,,    12.0000,    36.0000,     0.0000,
46 ,,,,    18.0000,    36.0000,     0.0000,
47 ,,,,    24.0000,    36.0000,     0.0000,
48 ,,,,    30.0000,    36.0000,     0.0000,
49 ,,,,    36.0000,    36.0000,     0.0000,
50 ,,,,    42.0000,    36.0000,     0.0000,
51 ,,,,    48.0000,    36.0000,     0.0000,
52 ,,,,     0.0000,    42.0000,     0.0000,
53 ,,,,    12.0000,    42.0000,     0.0000,
54 ,,,,    24.0000,    42.0000,     0.0000,
55 ,,,,    36.0000,    42.0000,     0.0000,
56 ,,,,    48.0000,    42.0000,     0.0000,
57 ,,,,     0.0000,    48.0000,     0.0000,
58 ,,,,     6.0000,    48.0000,     0.0000,
59 ,,,,    12.0000,    48.0000,     0.0000,
60 ,,,,    18.0000,    48.0000,     0.0000,
61 ,,,,    24.0000,    48.0000,     0.0000,
62 ,,,,    30.0000,    48.0000,     0.0000,
63 ,,,,    36.0000,    48.0000,     0.0000,
64 ,,,,    42.0000,    48.0000,     0.0000,
65 ,,,,    48.0000,    48.0000,     0.0000,
```

```
*MATERIAL
 EX    ,       1  ,  ,   0.10500E+08
 NUXY  ,       1  ,  ,   0.33300E+00
 DENS  ,       1  ,  ,   0.25879E-03
*I2
5,0,20,0,0.0,0.0,1.0E-5
*SPDISP
** SET ID =        1
       1,ROTY,     0.000000
       1,ROTX,     0.000000
       1,UZ   ,    0.000000
       1,UY   ,    0.000000
       1,UX   ,    0.000000
       2,ROTY,     0.000000
       2,UZ   ,    0.000000
       2,UY   ,    0.000000
       2,UX   ,    0.000000
       3,ROTY,     0.000000
       3,UZ   ,    0.000000
       3,UY   ,    0.000000
       3,UX   ,    0.000000
       4,ROTY,     0.000000
       4,UZ   ,    0.000000
       4,UY   ,    0.000000
       4,UX   ,    0.000000
       5,ROTY,     0.000000
       5,UZ   ,    0.000000
       5,UY   ,    0.000000
       5,UX   ,    0.000000
       6,ROTY,     0.000000
       6,UZ   ,    0.000000
       6,UY   ,    0.000000
       6,UX   ,    0.000000
       7,ROTY,     0.000000
       7,UZ   ,    0.000000
       7,UY   ,    0.000000
       7,UX   ,    0.000000
       8,ROTY,     0.000000
       8,UZ   ,    0.000000
       8,UY   ,    0.000000
       8,UX   ,    0.000000
       9,ROTY,     0.000000
       9,ROTX,     0.000000
       9,UZ   ,    0.000000
       9,UY   ,    0.000000
       9,UX   ,    0.000000
      10,ROTX,     0.000000
      10,UZ   ,    0.000000
      10,UY   ,    0.000000
      10,UX   ,    0.000000
      11,UY   ,    0.000000
      11,UX   ,    0.000000
      12,UY   ,    0.000000
      12,UX   ,    0.000000
      13,UY   ,    0.000000
      13,UX   ,    0.000000
      14,ROTX,     0.000000
      14,UZ   ,    0.000000
      14,UY   ,    0.000000
      14,UX   ,    0.000000
```

```
15,ROTX,     0.000000
15,UZ   ,    0.000000
15,UY   ,    0.000000
15,UX   ,    0.000000
16,UY   ,    0.000000
16,UX   ,    0.000000
17,UY   ,    0.000000
17,UX   ,    0.000000
18,UY   ,    0.000000
18,UX   ,    0.000000
19,UY   ,    0.000000
19,UX   ,    0.000000
20,UY   ,    0.000000
20,UX   ,    0.000000
21,UY   ,    0.000000
21,UX   ,    0.000000
22,UY   ,    0.000000
22,UX   ,    0.000000
23,ROTX,     0.000000
23,UZ   ,    0.000000
23,UY   ,    0.000000
23,UX   ,    0.000000
24,ROTX,     0.000000
24,UZ   ,    0.000000
24,UY   ,    0.000000
24,UX   ,    0.000000
25,UY   ,    0.000000
25,UX   ,    0.000000
26,UY   ,    0.000000
26,UX   ,    0.000000
27,UY   ,    0.000000
27,UX   ,    0.000000
28,ROTX,     0.000000
28,UZ   ,    0.000000
28,UY   ,    0.000000
28,UX   ,    0.000000
29,ROTX,     0.000000
29,UZ   ,    0.000000
29,UY   ,    0.000000
29,UX   ,    0.000000
30,UY   ,    0.000000
30,UX   ,    0.000000
31,UY   ,    0.000000
31,UX   ,    0.000000
32,UY   ,    0.000000
32,UX   ,    0.000000
33,UY   ,    0.000000
33,UX   ,    0.000000
34,UY   ,    0.000000
34,UX   ,    0.000000
35,UY   ,    0.000000
```

```
35,UX   ,     0.000000
36,UY   ,     0.000000
36,UX   ,     0.000000
37,ROTX ,     0.000000
37,UZ   ,     0.000000
37,UY   ,     0.000000
37,UX   ,     0.000000
38,ROTX ,     0.000000
38,UZ   ,     0.000000
38,UY   ,     0.000000
38,UX   ,     0.000000
39,UY   ,     0.000000
39,UX   ,     0.000000
40,UY   ,     0.000000
40,UX   ,     0.000000
41,UY   ,     0.000000
41,UX   ,     0.000000
42,ROTX ,     0.000000
42,UZ   ,     0.000000
42,UY   ,     0.000000
42,UX   ,     0.000000
43,ROTX ,     0.000000
43,UZ   ,     0.000000
43,UY   ,     0.000000
43,UX   ,     0.000000
44,UY   ,     0.000000
44,UX   ,     0.000000
45,UY   ,     0.000000
45,UX   ,     0.000000
46,UY   ,     0.000000
46,UX   ,     0.000000
47,UY   ,     0.000000
47,UX   ,     0.000000
48,UY   ,     0.000000
48,UX   ,     0.000000
49,UY   ,     0.000000
49,UX   ,     0.000000
50,UY   ,     0.000000
50,UX   ,     0.000000
51,ROTX ,     0.000000
51,UZ   ,     0.000000
51,UY   ,     0.000000
51,UX   ,     0.000000
52,ROTX ,     0.000000
52,UZ   ,     0.000000
52,UY   ,     0.000000
52,UX   ,     0.000000
53,UY   ,     0.000000
53,UX   ,     0.000000
54,UY   ,     0.000000
54,UX   ,     0.000000
```

```
      55,UY   ,      0.000000
      55,UX   ,      0.000000
      56,ROTX,       0.000000
      56,UZ   ,      0.000000
      56,UY   ,      0.000000
      56,UX   ,      0.000000
      57,ROTY,       0.000000
      57,ROTX,       0.000000
      57,UZ   ,      0.000000
      57,UY   ,      0.000000
      57,UX   ,      0.000000
      58,ROTY,       0.000000
      58,UZ   ,      0.000000
      58,UY   ,      0.000000
      58,UX   ,      0.000000
      59,ROTY,       0.000000
      59,UZ   ,      0.000000
      59,UY   ,      0.000000
      59,UX   ,      0.000000
      60,ROTY,       0.000000
      60,UZ   ,      0.000000
      60,UY   ,      0.000000
      60,UX   ,      0.000000
      61,ROTY,       0.000000
      61,UZ   ,      0.000000
      61,UY   ,      0.000000
      61,UX   ,      0.000000
      62,ROTY,       0.000000
      62,UZ   ,      0.000000
      62,UY   ,      0.000000
      62,UX   ,      0.000000
      63,ROTY,       0.000000
      63,UZ   ,      0.000000
      63,UY   ,      0.000000
      63,UX   ,      0.000000
      64,ROTY,       0.000000
      64,UZ   ,      0.000000
      64,UY   ,      0.000000
      64,UX   ,      0.000000
      65,ROTY,       0.000000
      65,ROTX,       0.000000
      65,UZ   ,      0.000000
      65,UY   ,      0.000000
      65,UX   ,      0.000000
*MODEO
1,1,1,1
*EIGO
1,0,3,1,0,1
*ENDDATA
```

COMMAND>GRD,ADD,1,0/0/0

This adds a grid (a point in 3-D space), calls it grid #1, and puts it at coordinates (0,0,0).

COMMAND>GRD,TRS,1,2,48

This makes grid #2 by "translating" grid #1 48 units in the X direction. Actually it translates grid #1 48/0/0 units where the format is X/Y/Z: since there are no "0"s typed in, they are automatically assigned. If it was desired that #2 be placed 48 units in the Y direction, the input would be either "GRD,TRS,1,2,0/48" or "GRD,TRS,1,2,/48". Default values are "0".

COMMAND>LIN,2GD,1,2,1

This creates a line starting at grid #1 and ending at grid #2, calling it line #1. The format is "LIN,2GD,starting grid #,ending grid #".

COMMAND>LIN,TRS,1,2,0/48

This creates line #2 by translating line #1 48 units in the Y direction. This command could be either be input as is or as "LIN,TRS,1,2,/48". The format is

"LIN,TRS,3,4,5A/5B/5C" where:

Entry 3 in the line to be translated.

Entry 4 is the line # to be created.

Entry 5 is the direction (X/Y/Z) entry 3 is to be translated in. Default values in this entry are "0"s.

COMMAND>PAT,2LN,1,2,1

PAT stands for patch, which is defined as a plate of defined size in 2 D space or a surface of defined size in 3 D space.. This entry creates patch #1 between lines #1 and #2 with the following format: "PAT,2LN,LINE ID #1, LINE ID #2, PATCH #". Refer to the example at the end of verification problem break down for an example of how a is created and what it looks like.

COMMAND>FEG,QUA,1,,,2/4/4/20

FEG stands for "Finite Element Generation". This entry creates a group of 16 quadrilateral shell elements, 4 elements by 4 elements, in the plate created by the above patch. This is accomplished with the following format: "FEG,QUA,3,4,5,6A/6B/6C/6D,7A/7B/7C/7D where:

Entry 3 is the patch ID number that is to be "FEGED".

Entry 4 is element ID list. This is the list of

the element numbers to be created. If left blank the program will automatically number the elements being created.

Entry 5 is the list of node numbers to be created. Like the element entry above if this entry is left blank the program will automatically number the nodes.

Entry 6A is the order of the elements to be created. In this case the elements are second order shell elements, of NKTP type 20, and contain 8 nodes per element. Refer to the Element Section of the DISPLAY II User's Manual.

Entry 6B and 6C are the number of elements to be generated along the parametric directions C1 and C2. Parametric directions are internally assigned by the program to two edges of a patch and three edges of a hyperpatch and are used by the program for defining mesh density of the patch. For example suppose it is desired to "break" a line 1/3 of the way from the end. Where is the end and where is the start? The program uses parametric directions to show where the start (where the directions intersect) and end is. Refer to pages 2.2-7 to 2.2-10 of the DISPLAY II User's Manual.

Entry 6D is the NKTP #, which stands for the elements type identification number. Each

element type has a different NKTP #. For an explanation of exactly what a specific NKTP # is, refer to the element library section of the NISA II User's Manual, pages 4.0-1 to 4.120-5.

Entry 7A and 7B are the material and property ID #'s respectfully. This set of #s, assigned to one or more elements, is used later in the input phase for assigning actual properties. If left blank the program automatically assigns a material ID # of 1 and a property ID # of 1. In this case all the elements have the same material characteristics and property characteristics and therefore all have the same MID # and the same PID #.

Entry 7C and 7D are the zoom values to be assigned along the parametric directions discussed in entry 6B and 6C above. If left blank the program assigns zoom = 1 to both.

COMMAND>PROP,ADD,1,1/1/1/1/1/1/1/1

This adds a thickness of 1 unit to all eight nodes of the elements with PID = 1 with the following format "PROP,ADD,3,4A/4B/4C..." where:

Entry 3 is the property index # (PID).

Entry 4 is the list of properties which are defined by the element type with each different

property separated by a /. Refer to the specific element type in the element library section of the NISA II User's Manual for more info on what properties are listed here.

COMMAND>MAT,ADD,1,10.5E6/0.3333/////////2.5879E-4

This adds the material properties of modulus of elasticity (EX) = 10.5E6 psi, Poisson's ratio (NUXY) = 0.333 and mass density = 2.5879E-4 lbs.-sec squared per inches to the fourth power to all the elements with material ID #1. This is accomplished with the following format: MAT,ADD,3,4A/4B/4C..." where:

Entry 3 is the material ID #.

Entry 4 is the list of material properties. Like the property characteristics described above, this list will vary for each element type with each separate property being separated by a slash "/".

COMMAND>DISP,ADD,1/9/57/65,1,0/0/0/0/0

This constrains nodes 1,9,57, and 65 in the X, Y, Z, Rotation in X and Rotation in Y, and assigns a constraint ID # of 1 with the following format: "DISP,ADD,3A/3B/3C...,4,5A-5F", where:

Entry 3 is the list of nodes to be constrained.

Entry 4 is the constraint ID #. The set ID under which the constraints are specified. Entry 5A to 5F are the degree(s) of freedom that are constrained. This is accomplished by placing a "0" in the degree of freedom that must be constrained with the following format: "X/Y/Z/ROTX/ROTY/ROTZ". Consecutive slashes or constrained directions not listed, indicate a null entry which the program either ignores or places a default value in that place. In this case the program ignores the null entries. For more information on null entries refer to the general command rules previously addressed in this chapter of to pages 2.1-3 to 2.1-6 of the DISPLAY II User's Manual.

COMMAND>DEC

This places the program in the mode to accept Executive Commands. For more info on executive commands refer to the previous Executive command overview and page 7.10-1 of the DISPLAY II User's Manual.

COMMAND>INS

This places the program in the mode to allow the insertion of data lines.

COMMAND>ANAL=EIGEN

This tells the program that the analysis to be performed is an EIGENVALUE analysis.

COMMAND>FILE=E01

This provides the file name prefix E01 for the internally generated binary data files.

COMMAND>SAVE=26,27

This tells the program to save the internally generated binary data files 26 and 27. The names of these files will be E0126.dat and E0127.dat respectively. Note this command must be used to save the data files 26 and 27, which are required for post-processing or dynamic analysis.

COMMAND>RESE=ON

This command tells the program to look at the numbering of the elements and resequence them (ie renumber them internally) to make the finite element analysis more efficient. The input of this command is not required as the program will automatically perform this function. When creating a finite element model there is definite "pecking order" of how elements and nodes should be

numbered. The analysis of a model that does not follow this order will take longer than one that does. This command has the program check the model to see if it conforms to the "pecking order". If the model does not then the program will internally change the element numbering so that it does. The change will not effect the user as it is transparent to the user and the element IDs that the user uses are not changed. For more info refer page 5.3-5 of the NISA II User's Manual

COMMAND>AUTO=ON

This command activates the automatic spurious normal rotation constraint for shells. This input is not specifically required as the default mode is for this to be on. Nodes in basic shell theory have 5 degrees of freedom. When transferring a model from the non-global system to a global system rotation in the theta Z direction may occur. Additionally rotation perpendicular to the theta Z direction may occur. If this happens then the shell (ie model) will not be defined and therefore cannot be solved. Having the AUTO CONSTRAINT on will constrain this perpendicular rotation, thereby allowing the model to be defined.

COMMAND>EIGE=SUBS,CONV

This command tells the program that the eigenvalue extraction method is to be conventional subspace. This command produces some effect on the speed of the analysis. If the eigenvalues are far apart then having the method be accelerated subspace will cause the program to aggressively shift the frequencies in order to speed up the convergence onto the natural frequencies. Refer to pages 3.5-1 to 3.5-4 and 5.3-6 in the NISA II User's Manual.

COMMAND>MASS=LUMPED

This command tells the program to activate the lumped matrix formulation, ie it tells the program to solve the stiffness matrix diagonally instead of consistent with the way the matrix was developed. Refer to page 3.5-5 of the NISA II User's Manual.

COMMAND>*LDCASE,1

This is the load case "control card" for the desired output. Refer to page 7.1-2 of the NISA II User's Manual for an explanation of what the number(s) following the card heading mean. Note when performing an Eigenvalue analysis this card is not applicable. <u>However, using this card was the only method the author found of "saving" the</u>

displacement constraints that were input previously. A more difficult method of inputing the displacement constraints would be to input them directly into the NIS file using a text or line editor, like EDLIIN. Therefore the user should either use this command to save the displacement constraints and then delete the two specific load case lines in the NIS file by using a text or line editor, like EDLIN, or input the displacement constraints directly into the NIS file. (The NIS file will be explained below.) The load case lines that need to be deleted consists of one line containing *LDCASE and a second line containing the string of numbers input immediately after the word *LDCASE above.

COMMAND>*EIGCNTL,5,0,20,0,0.0,0.0,1.0E-5

This command tells the program what the control parameters are for the eigenvalue extraction. Refer to pages 7.1-5 to 7.1-6a of the NISA II User's Manual for an explanation of what each number means.

COMMAND>*MODEO,1,1,1,1

This command tells the program what modes the analysis is required on. The format is "MODEO,2,3,4,5" where:

Entry 2 is the first mode number to be

analyzed.

Entry 3 is the last mode number to be analyzed.

Entry 4 is the increment between the first and last modes that are to be analyzed.

Entry 5 is the output ID # that refers to the ID # listed in the output command *EIGO.

For more info refer to page 7.1-15 of the NISA II User's Manual.

COMMAND>*EIGO,1,1,0,3,1,0,1

This command tells the program what output options to provide. For an explanation of what each number means refer to pages 7.5-2 to 7.5-4 of the NISA II User's Manual.

Note: there seems to be a problem with the MODEO and EIGO commands being accepted by the DISPLAY-PRE program and then saved in the NIS. file. As a result the above commands must be inputed directly into the ".NIS" file using either a text or line editor. This is accomplished with the following format:

line x: *MODEO

line x+1: modeo data

line x+2: *EIGO

line x+3: eigo data

where line x+4 is the ENDDATA line

The EIGO line must be the first output line listed. If any other output is requested (Example REGIONS, the regions for averaged nodal stresses printout; STRSFIL, the nodal stress filtering data; etc.; it must be listed after the EIGO data line. Refer to Section 7.5 OUTPUT CONTROL DATA of the NISA II User's Manual.

COMMAND>QUI

Quits the insert mode.

COMMAND>QUI

Quits the DEC mode.

COMMAND>WRI,NISA,E01.NIS

This takes the above data just input during this session and writes a "NISA" file called "E01.NIS" that the program uses for it's analysis.

COMMAND>WRI,DBS,E01.BIN

This takes the above data and writes a binary data base file called "E01.BIN"

```
COMMAND>END
```

This ends the session.

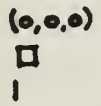
Data Base File: At this time the program will ask if a data base file DISP.BIN is to be written. Since this has already been done with the WRI,DBS,E01.BIN command above, answer no "N".

The model is now ready to be analyzed. Refer to the previous section "Program Usage" for guidance on how to perform the analysis.

COMMAND INPUT DEMONSTRATION

In an attempt to assist the unfamiliar user with understanding what the various DISPLAY-PRE commands actually do, specifically the creation of a patch, the demonstration below provides a pictorial description of some of the various commands required in the previous verification problem. This demonstration does not contain all required commands only a few to provide an idea of how the commands work.

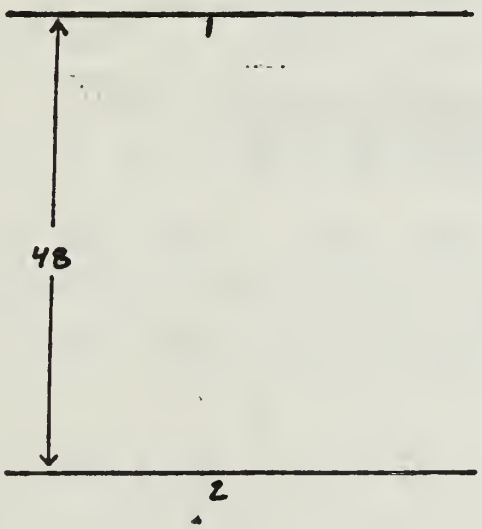
GRD,ADD,1,0/0/0 This creates a point at coordinates (0,0,0).



GRD,TRS,1,2,48 This creates point (grid) #2 by translating grid #1 48 units in the X direction.

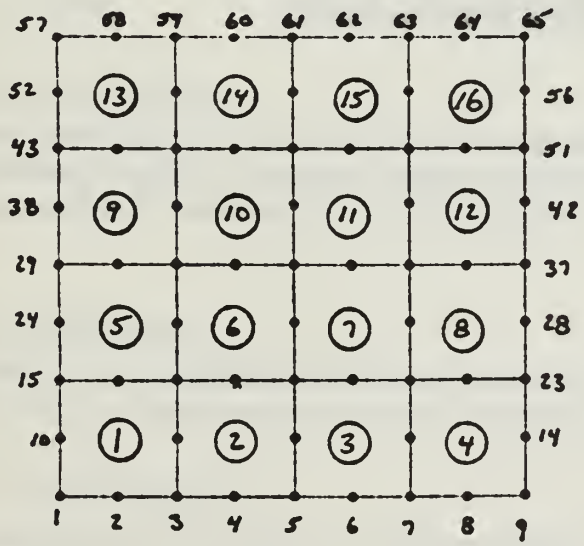
LIN,2GD,1,2,1 This creates line #1 between grids #1 and #2.

LIN,TRS,1,2,0/48 This creates line #2 by translating line #1 48 units in the Y direction.



PAT,2LN,1,2,1 This creates patch #1 between lines #1 and #2.

FEG,QUA,1,,,2/4/4/20 This creates a group of 16 quadrilateral shell elements of NORDR 2, 4 elements by 4 elements, in the patch created above. Due to space limitation not all the nodes are numbered.



VERIFICATION PROBLEM 6.3

TITLE:

Transient response of a beam subjected to distributed load

ELEMENT TYPE:

2-D beam element (NKTP = 13, NORDR = 1)

2-D translational spring element (NKTP = 18)

PROBLEM:

A beam supported by two springs of stiffness $K/2 = \pi^4 EI/2L^3$ at its ends, as shown in Fig. 6.3.1, is subjected to a distributed load given by:

$$p(x) = 1\text{-}2\,|x|/L \qquad \text{for } -L/2 \le x \le L/2$$

The variation of the load with time is shown in Fig. 6.3.2. Lateral motion of the beam is constrained. The natural frequencies and time history response of the mid-point of the beam are to be computed.

PROPERTIES:

Material:

EX	=	3.0×10^7 psi	(Modulus of elasticity)
NUXY	=	0.3	(Poisson's ratio)
DENS	=	0.000728 lbs.-sec^2 /in^4	(Mass density)

Cross-Section:

A	=	4.0 in^2	(Cross-sectional area)
IZZ	=	1.3333 in^4	(Moment of inertia about beam Z-axis)

FINITE ELEMENT MODEL:

The finte element model is shown in Fig. 6.3.3. Since the applied load and the system are symmetric, only one half of the beam is modeled. Symmetry boundary conditions (UX = 0.0, ROTZ = 0.0) are applied at node 1. Lateral motion is constrained (UX = 0.0) at nodes 2 and 3, and node 4 is fixed (UX = 0.0, UY = 0.0).

SOLUTION PROCEDURE:

Consistent mass matrix formulation and subspace iteration technique is used in evaluating the natural frequencies. First two symmetric modes are used for the transient dynamic analysis.

RESULTS & COMPARISON:

The natural frequencies of the first two symmetric modes are compared to the theoretical values in Table 6.3.1. The peak transient response of the mid-point of the beam is compared to the theoretical value in Table 6.3.2. The displacement history of the mid-point of the beam is shown in Fig. 6.3.4.

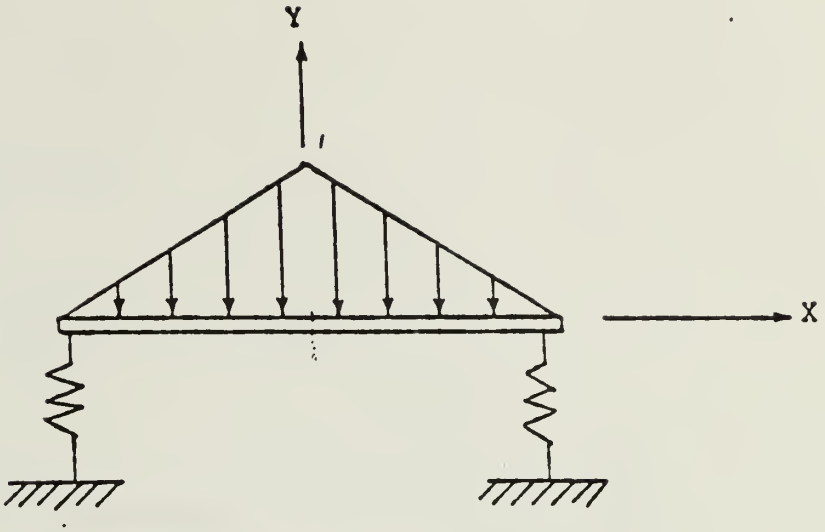


Figure 6.3.1 Beam with Distributed Load

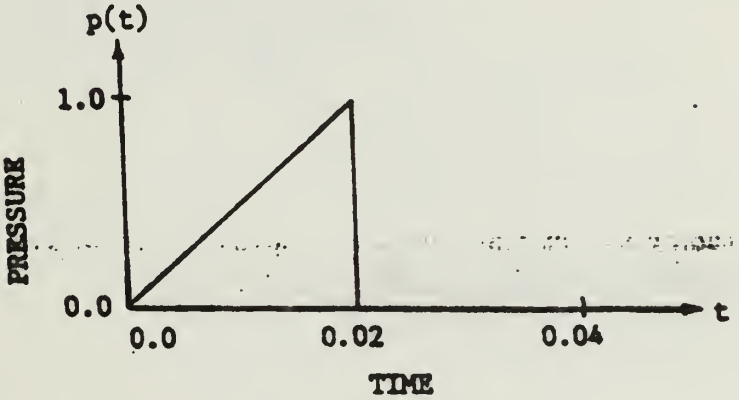


Figure 6.3.2 Load-time History

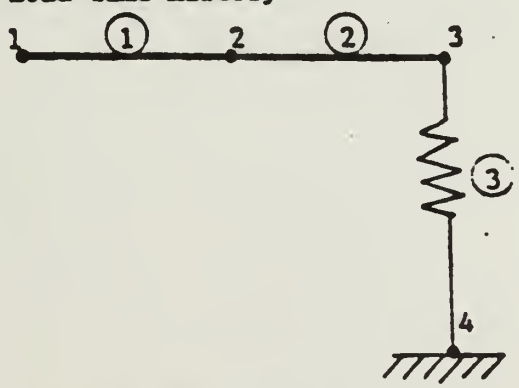


Figure 6.3.3 Finite Element Model

```
GRD,ADD,1,0/0/0
GRD,TRS,1,2,40
LIN,2GD,1,2,1
FEG,BAR,1,,,2/13,1/1
NOD,ADD,4,40/-4
ELE,ADD,3,18/1//2,3/4
DISP,ADD,1,1,0/////0
DISP,ADD,2T3,1,0
DISP,ADD,4,1,0/0
PROP,ADD,1,4./1.3333
MAT,ADD,1,3.0E7/0.3/////////7.28E-4
PROP,ADD,2,3.805E3
DEC
INS
ANAL=EIGENVALUE
AUTOCONSTRAINT=OFF
RESEQUENCING OF ELEMENTS=ON
FILE=ETD01
SAVE=26,27
EIGEN EXTRACTION=SUBSPACE
MASS MATRIX FORMULATION=CONSISTENT
*TITLE,ATTEMPT OF VERIFICATION PROBLEM ET03.STD
*I2,5,-1,20,0,0.0,0.0,1.0E-5
*I3,1
QUI
QUI
WRI,NISA,ETD01.NIS
WRI,DBS,ETD01.BIN
END
N
```

```
** EXECUTIVE
ANAL=EIGENVALUE
AUTOCONSTRAINT=OFF
RESEQUENCING OF ELEMENTS=ON
FILE=ETD01
SAVE=26,27
EIGEN EXTRACTION=SUBSPACE
MASS MATRIX FORMULATION=CONSISTENT
*TITLE
ATTEMPT OF VERIFICATION PROBLEM ET03.STD
*ELTYPE
  1,  13,      1
  2,  18,      1
*RCTABLE
       1,      2
   4.0000,    1.3333,
       2,      1
3805.0000,
*ELEMENT
       1,      1,      1,      1, 0
       1,      2,
       2,      1,      1,      1, 0
       2,      3,
       3,      0,      2,      2, 0
       3,      4,
*NODES
       1 ,,,,     0.0000,     0.0000,     0.0000,
       2 ,,,,    20.0000,     0.0000,     0.0000,
       3 ,,,,    40.0000,     0.0000,     0.0000,
       4 ,,,,    40.0000,    -4.0000,     0.0000,
*MATERIAL
 EX    ,      1 , ,  0.30000E+08
 NUXY  ,      1 , ,  0.30000E+00
 DENS  ,      1 , ,  0.72800E-03
*I2
5,-1,20,0,0.0,0.0,1.0E-5
*SPDISP
** SET ID =       1
       1,ROTZ,     0.000000
       1,UX   ,    0.000000
       2,UX   ,    0.000000
       3,UX   ,    0.000000
       4,UY   ,    0.000000
       4,UX   ,    0.000000
*ENDDATA
```

```
COMMAND>GRD,ADD,1,0/0/0
```

This adds a grid (a point in 3-D space), calls it grid #1, and puts it at coordinates (0,0,0).

```
COMMAND>GRD,TRS,1,2,40
```

This makes grid #2 by "translating" grid #1 40 units in the X direction. Actually it translates grid #1 40/0/0 units where the format is X/Y/Z: since there are no "0"s typed in, they are automatically assigned. If it was desired that #2 be placed 40 units in the Y direction, the input would be either "GRD,TRS,1,2,0/40" or "GRD,TRS,1,2,/40". Default values are "0".

```
COMMAND>LIN,2GD,1,2,1
```

This creates a line starting at grid #1 and ending at grid #2, calling it line #1. The format is "LIN,2GD,starting grid #,ending grid #".

```
COMMAND>FEG,BAR,1,,,2/13,1/1
```

FEG stands for "Finite Element Generation". This entry creates a group of 2 bar/beam elements (of element type 13), in the global coordinate system, assigning a

material ID # of 1 and a property ID # of 1, and a zoom of 1. Each element is defined with a node on each end and both nodes and elements are automatically numbered by the program. This is accomplished using the following format: "FEG,BAR,3,4,5,6A/6B/6C,7A/7B/7C" where:

Entry 3 is the number of lines to be "FEGED".

Entry 4 is the list of generated elements. If left blank the program will automatically number the elements.

Entry 5 is the list of generated nodes. If left blank the program will automatically number the nodes.

Entry 6A is the number of elements to be generated for each line.

Entry 6B is the NKTP #, which stands for the element type identification number. Each element type has a different NKTP #. For an explanation of exactly what a specific NKTP # is, refer to the element library section of the NISA II User's Manual, pages 4.0-1 to 4.120-5.

Entry 6C is the node # that has a local coordinate system the user may wish to use. This entry is used only if the beam is not to be defined in the global coordinate system (ie the Cartesian, Cylindrical, or Spherical systems). At times it is more convenient to refer to items with a local coordinate system, for example a

force or displacement constraint may be defined with directions which are not aligned with the global coordinate system. For more information on local coordinate systems refer to pages 3.2-1 to 3.2-4 of the NISA II User's Manual.

Entry 7A & 7B are the material and property ID #'s respectfully. This set of #s, assigned to one or more elements, is used later in the input phase for assigning actual properties. If left blank the program automatically assigns a material ID # of 1 and a property ID # of 1. These numbers do not have to be the same. A group of elements can have the same material properties (Poisson's ratio, Young's modulus, etc.) with some of the elements having one geometric property and others having a different geometric property. As a result this group of elements would have one material ID # and two property ID #s.

Entry 7C is the zoom factor, ie the length of the last element to length of the first element. If all elements are the same length then the zoom = 1. If left blank the program automatically assigns a value of 1. For more information and a pictorial representation refer to page B-3 of the DISPLAY II User's Manual.

COMMAND>NOD,ADD,4,40/-4

This command adds a node at coordinates (40,-4,0) and calls it node # 4. This is accomplished with the following format: "NOD,ADD,3,4A/4B/4C/4D" where:

Entry 3 is the node # to be generated.

Entry 4A, 4B, & 4C are the (X/Y/Z) coordinates of the node.

Entry 4C is the local coordinate system in which the coordinates are to be specified. If this is left blank then the node is in the global system.

COMMAND>ELE,ADD,3,18/1//2,3/4

This adds an element #3 with NKTP = 18, order = 1, property ID # = 2 between nodes 3 and 4, using the following format: "ELE,ADD,3,4A/4B/4C/4D,5A/5B,6" where:

Entry 3 is the element # to be added. If this is left blank the program will automatically assign a number.

Entry 4A is the NKTP # (see entry 6B under the FEG description above).

Entry 4B is the order #. Some elements only have one order, NORDR=1, others have many depending on the element type. For a pictorial description of how this order works/varies refer to section 4, the "element library" of the NISA

II User's Manual; specifically NKTP #s 1,2,3,4,5,& 32 provide good examples of some of the element types with varying orders.

Entry 4C is the material ID #. This # is used later in the input phase to assign actual material properties to this element.

Entry 4D is the property ID #. (See entry 7B under the FEG description above.)

Entry 5A & 5B are the two nodes that define the element. Most elements require two nodes, one at each end. However since this particular element is a mass concentration it only requires one node.

Entry 6 is the vector number of a previously defined vector used in establishing a local coordinate system. This can be used for elements of NKTP = 11, 12, & 38.

```
COMMAND>DISP,ADD,1,1,0/////0
```

This constrains node #1 in the X and ROTZ directions and assigns a constraint ID # of 1 with the following format: "DISP,ADD,3A/3B/3C...,4,5A-5F", where:

Entry 3 is the list of nodes to be constrained.

Entry 4 is the constraint ID #. The set ID under which the constraints are specified.

Entry 5A to 5F are the degree(s) of freedom that

are constrained. This is accomplished by placing a "0" in the degree of freedom that must be constrained with the following format: "X/Y/Z/ROTX/ROTY/ROTZ". Consecutive slashes or constrained directions not listed, indicate a null entry which the program either ignores or places a default value in that place. In this case the program ignores the null entries. For more information on null entries refer to the general command rules previously addressed in this chapter of to pages 2.1-3 to 2.1-6 of the DISPLAY II User's Manual.

COMMAND>PROP,ADD,1,4./1.3333

This adds a cross sectional area of 4.0 inches squared and a moment of inertia on 1.3333 inches to the forth to all the elements with property ID # 1. This is accomplished with the following format "PROP,ADD,3,4A/4B/4C..." where:

Entry 3 is the property index # (PID).

Entry 4 is the list of properties which are defined by the element type with each different property separated by a /. Refer to the specific element type in the element library section of the NISA II User's Manual for more info on what properties are listed here.

COMMAND>MAT,ADD,1,3.0E7/0.3/////////7.28E-4

This adds the material properties of Modulus of elasticity (EX) = 3.0E7 psi, Poisson's ratio (NUXY) = 0.3, and mass density (DENS) = 7.28E-4 lbs.-sec squared per inches to the forth. This is accomplished with the following format:" MAT,ADD,3,4A/4B/4C/..." where:

Entry 3 is the material ID st #.

Entry 4 is the list of material properties. Like the property characteristics described above, this list will vary for each element type.

COMMAND>DEC

This places the program in the mode to accept Executive Commands. For more info on executive commands refer to the previous Executive command overview and page 7.10-1 of the DISPLAY II User's Manual.

COMMAND>INS

This places the program in the mode to allow the insertion of data lines.

COMMAND>ANAL=EIGENVALUE

This tells the program that the analysis to be performed is an EIGENVALUE analysis.

COMMAND>AUTOCONSTRAINT=OFF

This command deactivates the automatic spurious normal rotation constraint. Since these elements are bar/beam elements this command, which basically applies to shell elements, is not really applicable. However, the following explanation is provided for what this command does if it is activated. Nodes in basic shell theory have 5 degrees of freedom. When transferring a model from the non-global system to a global system rotation in the theta Z direction may occur. Additionally rotation perpendicular to the theta Z direction may occur. If this happens then the shell (ie model) will not be defined and therefore cannot be solved. Having the AUTO CONSTRAINT on will constrain this perpendicular rotation, thereby allowing the model to be defined.

COMMAND>RESEQUENCING OF ELEMENTS=ON

This command tells the program to look at the numbering of the elements and to resequence them (ie renumber them internally) to make the finite element analysis more efficient. This command is not really

required as the he program is set up to automatically perform this function. When creating a large finite element model this process (ie RESE=ON) is usually necessary because there is a definite "pecking order" of how elements and nodes should be numbered. The analysis of a model that does not follow this order will take longer than one that does. This command has the program check the model to see if it conforms to the "pecking order". If the model does not then the program will internally change the element numbering so that it does. The change will not effect the user as it is transparent to the user and the element IDs that the user uses are not changed. For more info refer page 5.3-5 of the NISA II User's Manual.

COMMAND>FILE=ETD01

This provides the file name prefix ETD01 for the internally generated binary data files.

COMMAND>SAVE=26,27

This tells the program to save the internally generated binary data files 26 and 27. The names of these files will be ETD0126.dat and ETD0127.dat respectively. Note this command must be used to save the data files 26 and 27, which are required for post-processing or dynamic

analysis.

COMMAND>EIGEN EXTRACTION=SUBSPACE

This command tells the program that the eigenvalue extraction method is to be accelerated subspace. The accelerated selection of the subspace type is the default setting and therefore the accelerated did not have to be specified. This command produces some effect on the speed of the analysis. If the eigenvalues are far apart then having the method be accelerated subspace will cause the program to aggressively shift the frequencies in order to speed up the convergence onto the natural frequencies. Refer to pages 3.5-1 to 3.5-4 and 5.3-6 in the NISA II User's Manual.

COMMAND>MASS MATRIX FORMULATION=CONSISTENT

This command tells the program to activate the consistent matrix formulation, ie it tells the program to solve the stiffness matrix consistent with the way the matrix was developed. Using a consistent mass matrix formulation usually gives more accurate results than the other method available, which is lumped. Refer to page 3.5-5 of the NISA II User's Manual.

COMMAND>*TITLE, ATTEMPT OF VERIFICATION PROBLEM ET03.STD

This command tells the program that the title of this analysis is "ATTEMPT OF VERIFICATION PROBLEM ET03.STD", which will be printed at the top of the answer output.

COMMAND>*I2,5,-1,20,0,0.0,0.0,1.0E-5

This command tells the program what the control parameters are for the eigenvalue extraction. Refer to pages 7.1-5 to 7.1-6a of the NISA II User's Manual for an explanation of what each number means.

COMMAND>*I3,1

*I3 is the same as *LDCASE which is the load case "control card" for the desired output. Refer to page 7.1-2 of the NISA II User's Manual for an explanation of what the number(s) following the card heading mean. Note when performing an Eigenvalue analysis this card is not applicable. However, using this card was the only method the author found of "saving" the displacement constraints that were input previously. A more difficult method of inputing the displacement constraints would be to input them directly into the NIS file using a text or line editor, like EDLIIN. Therefore the user should either use this command to save the displacement constraints and then delete the two specific load case lines in the NIS file by

using a text or line editor, like EDLIN, or input the displacement constraints directly into the NIS file. (The NIS file will be explained below.) The load case lines that need to be deleted consists of one line containing *LDCASE and a second line containing the string of numbers input immediately after the word *LDCASE above.

COMMAND>QUI

Quits the insert mode.

COMMAND>QUI

Quits the DEC mode.

COMMAND>WRI,NISA,ETD01.NIS

This takes the above data just input during this session and writes a "NISA" file called "ETD01.NIS" that the program uses for it's analysis.

COMMAND>WRI,DBS,ETD01.BIN

This takes the above data and writes a binary data base file called "ETD01.BIN"

COMMAND>END

This ends the session.

Data Base File: At this time the program will ask if a data base file DISP.BIN is to be written. Since this has already been done with the WRI,DBS,ETD01.BIN command above, answer no "N".

The model is now ready to have an eigenvalue analysis performed on it. Once this analysis has been completed a dynamic analysis can be preformed. Remember that files 26.dat and 27.dat must be saved during the eigenvalue analysis in order to be used to complete the dynamic analysis. Refer to the previous section "Program Usage" for guidance on how to perform the analysis.

```
ANAL=TRANSIENT
DAMP=VISCOUS
FILE=ETD01
STARTING TIME=0.0
DELTA TIME=0.0001
ENDTIME=0.08
*TITLE
ATTEMPT OF VERIFICATION PROBLEM TD03.STD
*MODESECTION
1,2,1,1
*DAMPING
1,0.0
*TIMEFUNCTION
LINEARLY INCREASING LOAD
1001,3,0,1.0
0.0,0.0,0.02,1.0,0.0201,0.0
*ARRIVALTIME
101,0.0
*DPRESSURE
1,1001,101,1,1
1.0,0.5
2,1001,101,1,1
0.5,0.0
*RSET
1001,1,UY
*HISTORY
0.0,0.08,1,1,1001
*ENDDATA
```

ANAL=TRANSIENT

This tells the program that the analysis to be performed is a transient dynamic analysis.

DAMP=VISCOUS

This tells the program that the model has viscous damping in it.

FILE=ETD01

This tells program that the prefix ETD01 is to be used for the internally generated binary data file. This prefix must be the same as that used for the eigenvalue portion of the problem. The reason for this is because the dynamic analysis must use the 26.dat and 27.dat files generated by the eigenvalue analysis, for the its' analysis. This command provides the prefix name to look for the required data.

STARTING TIME=0.0

This tells the program that the starting time of the forcing function is 0.0.

DELTA TIME=0.0001

This tells the program to analyze the response of the system from the starting time to the ending time in 0.0001 time increments. This value must be less than the difference between the starting time and the ending time or the program will not provide a response.

ENDTIME=0.08

This tells the program that the ending time of the forcing function is 0.08.

From this point on all commands will consists of more than one line of data. As a result a "*" indicates the start of a new set of data.

*TITLE

This is a two line data group. This line tells the program that the following line of input will be the analysis title.

ATTEMPT OF VERIFICÂTION PROBLEM TD03.STD

This is the title provided to the analysis which the program will then print at the top of the answer output.

*MODESECTION

This is a multi line data group. This line tells the program the following line(s) of input will contain the data specifying what modes are to be analyzed. This command is always required, refer to page 8.4-2 of the NISA II User's Manual.

1,2,1,1

This tells the program that modes 1 and 2, with an increment of 1, are to be analyzed with damping ID #1's damping value. This is accomplished with the following format: Entry 1,2,3,4 where:

Entry 1 is the first mode to be analyzed.

Entry 2 is the last mode to be analyzed.

Entry 3 is the increment between modes to be analyzed. Suppose the modes to be analyzed were modes 1, 5 and 10 then entry 1 would be 1, entry 2 would be 10 and entry 3 would be 2.

Entry 4 is the damping ID # that corresponds to the damping parameters set previously. The damping ID # allows a system to be analyzed for various damping values without remodeling the whole system each time. Suppose it was desired to analyze this problem with two different damping values, say 30% and 70% of critical. To

do this, two damping data groups would be required. Part of the damping data input is an arbitrary ID #. Let's let the 30% damping have an ID # of 1030 and the 70% have an ID # of 170 (note these numbers are totally arbitrary). Now to find the response of this system with those two values input:

```
*MODESECTION
1,2,1,1030
1,2,1,170
```

*TIMEFUNCTION

This is a multi line data group. This line tells the program that the following lines of input will provide the data used to specify the time function. This command is always required for transient dynamic analysis. Refer to page 8.5-3 of the NISA II User's Manual.

LINEARLY INCREASING LOAD

This is the title used to describe the time function.

1001,3,0,1.0

This tells the program that three points will be used to define the forcing function in terms of abscissa and ordinates, has a scale of 1.0, and a time function ID # of 1001. This is accomplished with the following format: Entry 1,2,3,$,4,5,6 where:

Entry 1 is the time function ID #. This is an arbitrary number that matches the number used in the *GROUND, *DPRESSURE and/or *DCFORCE data lines discussed later.

Entry 2 is the number of points used to define the time function.

Entry 3 provides the definition method. 0= the time function is defined in terms of abscissa and ordinates. 1= the time function is defined at equal intervals.

$ tells the program to skip all entries between the last one inputed and entry 4.

Entry 4 is the scale factor used for the time function. Basically there are two numbers that can be input here. The correct number depends on the definition of the time function. The easiest way to explain this is with an example. Therefore, let the time function = 200 sine 400 theta. Now if the amplitude was based on 200 sine 400 theta (ie theta = 0.5236 radians and

the force = 173.2) then the scale factor is 1. If the amplitude was based on sine 400 theta (ie theta = 0.5236 radians and the force = 0.866) then the scale factor = 200. However any number can be used as the scale factor as long as the user maintains consistency through out the modal.

Entry 5 is the starting time for the load function.

Entry 6 is the time increment used for the time function. If entry 3 is "0" this entry is not used.

0.0,0.0,0.02,1.0,0.0201,0.0

This defines the time function by using a maximum of three pairs of numbers per line, with the total number of pairs equal to the number listed in entry 2 above. The pairs of numbers consists of the first number referring to the time at point x and the second number of the pair is the function value at point x.

*ARRIVALTIME

This is two line data group that tells the program that the following line of input will specify arrival times used in ground motion, concentrated forces and/or

pressure data.

101,0.0

This tells the program that arrival time ID # 101 has an arrival time of 0.0. This is accomplished with entry 1 being an arbitrary number (referenced in the *GROUND, *DPRESSURE and/or *DCFORCE data lines) assigned as the ID # and entry 2 being the arrival time.

*DPRESSURE

This is a multi line data group. This line tells the program that the following lines of input will contain data specifying the pressure loading of elements.

1,1001,101,1,1

This tells the program that there is one additional line of input required to complete the pressure loading data, that the pressure is acting on face 1 of element 1, and the time function ID # is 1001 and the arrival time ID # is 101. This is accomplished with the following format: Entry 1,2,3,4,5,6,7,$,8 where:

Entry 1 is the element that the pressure is acting on.

Entry 2 is the time function ID #.

Entry 3 is the arrival time ID #. Note: The parameters that entries 2 and 3 refer to depend on the type of dynamic analysis being performed. For example if a frequency response analysis is to be performed entry 2 refers to the amplitude spectrum ID # and entry 3 refers to the phase spectrum ID #. Refer to pages 8.6-2 and 8.6-8 of the NISA II User's Manual.

Entry 4 is the face number that the pressure acts on.

Entry 5 is the number of additional lines required to complete the pressure loading data.

Entry 6 is the last element number that the pressure is acting on.

Entry 7 is the increment between entry 1 and entry 6 elements the pressure is acting on.

$ tells the program to skip all entries between the last one inputed and entry 8.

Entry 8 is the uniform pressure load intensity. This entry is used only if entry 5 = 0.

1.0,0.5

This tells the program that the pressure on the element referenced above (element #1) varies from a pressure of 1.0 on the left node to a pressure of 0.5 on the right node of the element. Refer to page 8.6-9 of the

*RSET

This is a multi line data group. This line tells the program that the following line(s) of input will define the nodal/element response quantities that are to be provided by the program. This command is required. Refer to page 8.7-2 of the NISA II User's Manual.

1001,1,UY

This tells the program that the response at node 1, in the Y direction, is to be provided with a response ID # of 1001. This is accomplished with the following format: Entry 1,2,3,4,5,$,6 where:

Entry 1 is an arbitrarily assigned response ID #.

Entry 2 is the node or element # to be analyzed.

Entry 3 is the response to be specifically provided.

Entry 4 is the last node or element # to be analyzed. If the response at only one node or element is required then this entry must be the same as entry 2. If not the program will state that there is an error in this data line and not provide a response.

Entry 5 is the node or element increment. If the response at only one node or element is required then with entry 4 equaling entry 2 this entry must be one "1". If not the program will state that there is an error in this data line and not provide a response.

$ This tells the program to skip all entries between the last one inputed and entry 6.

Entry 6 is the layer number for composite elements.

Note: Additional input lines can be used to specify responses for different nodes/elements and/or different responses. For example suppose the responses at node 1 in both the X and Y directions was desired. Then there would be two lines of input, one with entry 3 containing UX and one with entry 3 containing UY.

*HISTORY

This is a two line data group. This line tells the program that the following line of input will request the response history computation for printout and/or post processing. The NISA II User's Manual states that this command group is optional. However the author determined, by running various analyses, that this command is not optional. If this command and/or the *RSET command is not used then the program will have a "fatal" error and will

provide the following response "FATAL ERROR", "NO RESPONSE OUTPUT HAS BEEN REQUESTED".

0.0,0.08,1,1,1001

This tells the program that the time history, maximum values and time of occurrence of maximum values are to be printed starting at time 0.0, ending at time is 0.08, with increments of 1, and that the response ID # is 1001. This is accomplished with the following format:

Entry 1,2,3,4,$,5-14 where:

Entry 1 is the starting time. This entry must equal the value placed in the "STARTING TIME" Executive command.

Entry 2 is the ending time. This entry must equal the value placed in the "ENDTIME" Executive command.

Entry 3 is the history calculation increment. This increment must be smaller than the difference in time between the start time and the end time. If it is not the program will not provide a response.

Entry 4 is the printout option. 0 - response history calculated but not printed. 1 - time history, maximum values and time of occurrence of maximum values are printed. 2 - only maximum values and times of occurrence are printed.

$ tells the program to skip all entries between the last one inputed and entry 5. Entry 5 to 14 are the response set ID #s that the time history response is required. These numbers should be referenced in the *RSET input.

*ENDDATA

This tells the program that this is the last line of data.

Once this file is completed and the eigenvalue analysis has been performed the model is ready to have the dynamic analysis performed. To perform the dynamic analysis access the directory and sub-directory that the above file is saved in, type "NISA286" and hit the return. Now move the cursor, using the arrow keys, to the "DYNAMIC" selection and hit the return. At this time the program will ask for the input file name. Type in the name of the file above and hit the return. Now the program will ask for the output file name. Type in the desired output name and hit return. Now wait for the analysis.

VERIFICATION PROBLEM 8.2

TITLE:

Frequency response analysis of a 2-DOF system

ELEMENT TYPE:

2-D mass element (NKTP = 25)
2-D translational spring (NKTP = 18)

PROBLEM:

A 2DOF system (Fig. 8.2.1) is subjected to a harmonic force acting at node 2. Damping is neglected.

The response of the system is to be computed for an exciting frequency range of 0.0 to 7.5 HZ at intervals of 0.25 HZ.

PROPERTIES:

Material:
Translational mass = 200 lb. sec.^{2}/in.
Translational spring constant = 0.5 lb./in.

FINITE ELEMENT MODEL:

The finite element model consists of 4 active nodes and 5 elements (2 mass elements and 3 spring elements). Zero displacements are prescribed at the end points.

SOLUTION PROCEDURE:

An eigenvalue analysis using lumped mass formulation and conventional subspace iteration technique is carried out first, from which 2 eigenvalues and eigenvectors are obtained. The frequency response run is performed and displacement response (amplitude and phase lag) is computed.

RESULTS & COMPARISON:

The natural frequencies computed by NISA are compared with the theoretical results in Table 8.2.1. The displacement response results are presented in Table 8.2.2 along with theoretical results. The amplitude and phase spectra for the displacement at node 2 are shown in Figs. 8.2.2 and 8.2.3.

REFERENCE:

1. W.T. Thomson, "*Theory of Vibration with Applications*," Prentice Hall Inc., Englewood Cliffs, N.J., 2nd. edition (1981).

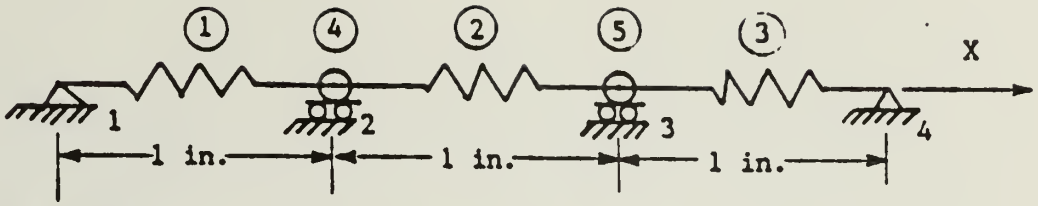


Figure 8.2.1 Two Degrees of Freedom System

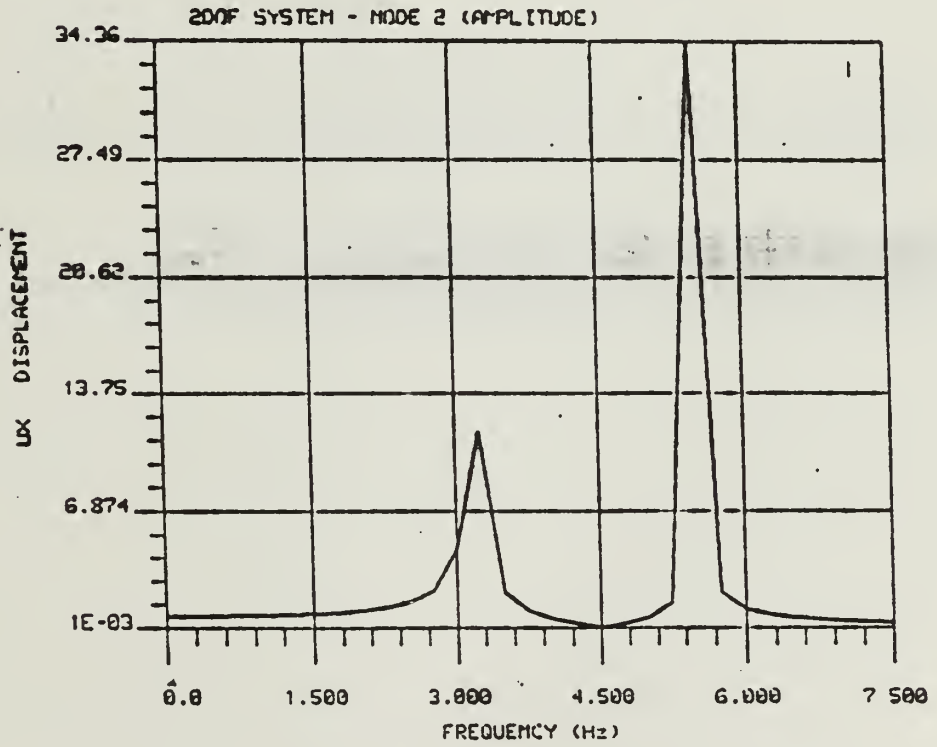


Figure 8.2.2 Response amplitude Spectrum for x-Displacement at node 2

```
GRD,ADD,1,0/0/0
GRD,TRS,1,2,3
LIN,2GD,1,2,1
FEG,BAR,1,,,3/18,1/1
ELE,ADD,4,25/1/2/2,2
ELE,ADD,5,25/1/2/2,3
DISP,ADD,1/4,1,0/0
DISP,ADD,2/3,1,/0
PROP,ADD,1,0.5
PROP,ADD,2,200.
DEC
INS
ANAL=EIGEN
FILE=EF01
SAVE=26,27
RESE=OFF
AUTO=ON
EIGE=SUBS,CONV
MASS=LUMPED
*LDCASE,1
*EIGCNTL,2,0,20,0,0.0,0.0,1.0E-05
QUI
QUI
WRI,NISA,EF01.NIS
WRI,DBS,EF01.BIN
END
N
```

```
** EXECUTIVE
ANAL=EIGEN
FILE=EF01
SAVE=26,27
RESE=OFF
AUTO=ON
EIGE=SUBS,CONV
MASS=LUMPED
*ELTYPE
  1,  18,      1
  2,  25,      1
*RCTABLE
       1,      1
200.0000
       2,      1
0.5000
*ELEMENT
       1,      1,      1,      1, 0
       1,      2,
       2,      1,      1,      1, 0
       2,      3,
       3,      1,      1,      1, 0
       3,      4,
       4,      2,      2,      2, 0
       2,
       5,      2,      2,      2, 0
       3,
*NODES
       1 ,,,,      0.0000,     0.0000,     0.0000,
       2 ,,,,      1.0000,     0.0000,     0.0000,
       3 ,,,,      2.0000,     0.0000,     0.0000,
       4 ,,,,      3.0000,     0.0000,     0.0000,
*I2
2,0,20,0,0.0,0.0,1.0E-05
*SPDISP
** SET ID =       1
       1,UY   ,     0.000000
       1,UX   ,     0.000000
       2,UY   ,     0.000000
       3,UY   ,     0.000000
       4,UY   ,     0.000000
       4,UX   ,     0.000000
*ENDDATA
```

```
COMMAND>GRD,ADD,1,0/0/0
```

This adds a grid (a point in 3-D space), calls it grid #1, and puts it at coordinates (0,0,0).

```
COMMAND>GRD,TRS,1,2,3
```

This makes grid #2 by "translating" grid #1 3 units in the X direction. Actually it translates grid #1 3/0/0 units where the format is X/Y/Z: since there are no "0"s typed in, they are automatically assigned. If it was desired that #2 be placed 3 units in the Y direction, the input would be either "GRD,TRS,1,2,0/3" or "GRD,TRS,1,2,/3". Default values are "0".

```
COMMAND>LIN,2GD,1,2,1
```

This creates a line starting at grid #1 and ending at grid #2, calling it line #1. The format is "LIN,2GD,starting grid #,ending grid #".

```
COMMAND>FEG,BAR,1,,,3/18,1/1
```

FEG stands for "Finite Element Generation". This entry creates a group of 3 bar/beam elements (of element type 18), in the global coordinate system, assigning a

material ID # of 1 and a property ID # of 1, and a zoom of 1. Each element is defined with a node on each end and both nodes and elements are automatically numbered by the program. This is accomplished using the following format: "FEG,BAR,3,4,5,6A/6B/6C,7A/7B/7C" where:

Entry 3 is the number of lines to be "FEGED".

Entry 4 is the list of generated elements. If left blank the program will automatically number the elements.

Entry 5 is the list of generated nodes. If left blank the program will automatically number the nodes.

Entry 6A is the number of elements to be generated for each line.

Entry 6B is the NKTP #, which stands for the element type identification number. Each element type has a different NKTP #. For an explanation of exactly what a specific NKTP # is, refer to the element library section of the NISA II User's Manual, pages 4.0-1 to 4.120-5.

Entry 6C is the node # that has a local coordinate system the user may wish to use. This entry is used only if the beam is not to be defined in the global coordinate system (ie the Cartesian, Cylindrical, or Spherical systems). At times it is more convenient to refer to items with a local coordinate system, for example a

force or displacement constraint may be defined with directions which are not aligned with the global coordinate system. For more information on local coordinate systems refer to pages 3.2-1 to 3.2-4 of the NISA II User's Manual.

Entry 7A & 7B are the material and property ID #'s respectfully. This set of #s, assigned to one or more elements, is used later in the input phase for assigning actual properties. If left blank the program automatically assigns a material ID # of 1 and a property ID # of 1. These numbers do not have to be the same. A group of elements can have the same material properties (Poisson's ratio, Young's modulus, etc.) with some of the elements having one geometric property and others having a different geometric property. As a result this group of elements would have one material ID # and two property ID #s.

Entry 7C is the zoom factor, ie the length of the last element to length of the first element. If all elements are the same length then the zoom = 1. If left blank the program automatically assigns a value of 1. For more information and a pictorial representation refer to page B-3 of the DISPLAY II User's Manual.

COMMAND>ELE,ADD,4,25/1/2/2,2

This adds an element #4 with NKTP = 25, order = 1, material ID # = 2, property ID # = 2 at node 2, using the following format: "ELE,ADD,3,4A/4B/4C/4D,5A/5B,6" where:

Entry 3 is the element # to be added. If this is left blank the program will automatically assign a number.

Entry 4A is the NKTP # (see entry 6B under the FEG description above).

Entry 4B is the order #. Some elements only have one order, NORDR=1, others have many depending on the element type. For a pictorial description of how this order works/varies refer to section 4, the "element library" of the NISA II User's Manual; specifically NKTP #s 1,2,3,4,5,& 32 provide good examples of some of the element types with varying orders.

Entry 4C is the material ID #. This # is used later in the input phase to assign actual material properties to this element.

Entry 4D is the property ID #. (See entry 7B under the FEG description above.)

Entry 5A & 5B are the two nodes that define the element. Most elements require two nodes, one at each end. However since this particular element is a mass concentration it only requires one node.

Entry 6 is the vector number of a previously defined vector used in establishing a local coordinate system. This can be used for elements of NKTP = 11, 12, & 38.

```
COMMAND>DISP,ADD,1/4,1,0/0
```

This constrains nodes 1 and 4 in the X and Y direction and assigns a constraint ID # of 1 with the following format: "DISP,ADD,3A/3B/3C...,4,5A-5F", where:

Entry 3 is the list of nodes to be constrained.

Entry 4 is the constraint ID #. The set ID under which the constraints are specified.

Entry 5A to 5F are the degree(s) of freedom that are constrained. This is accomplished by placing a "0" in the degree of freedom that must be constrained with the following format: "X/Y/Z/ROTX/ROTY/ROTZ". Consecutive slashes or constrained directions not listed, indicate a null entry which the program either ignores or places a default value in that place. In this case the program ignores the null entries. For more information on null entries refer to the general command rules previously addressed in this chapter of to pages 2.1-3 to 2.1-6 of the DISPLAY II User's Manual.

COMMAND>PROP,ADD,1,0.5

This adds a translational spring constant of 0.5 lb./in. to all the elements with property ID # 1. This is accomplished with the following format "PROP,ADD,3,4A/4B/4C..." where:

Entry 3 is the property index # (PID).

Entry 4 is the list of properties which are defined by the element type with each different property separated by a /. Refer to the specific element type in the element library section of the NISA II User's Manual for more info on what properties are listed here.

COMMAND>DEC

This places the program in the mode to accept Executive Commands. For more info on executive commands refer to the previous Executive command overview and page 7.10-1 of the DISPLAY II User's Manual.

COMMAND>INS

This places the program in the mode to allow the insertion of data lines.

COMMAND>ANAL=EIGEN

This tells the program that the analysis to be performed is an EIGENVALUE analysis.

COMMAND>FILE=EF01

This provides the file name prefix EF01 for the internally generated binary data files.

COMMAND>SAVE=26,27

This tells the program to save the internally generated binary data files 26 and 27. The names of these files will be EF0126.dat and EF0127.dat respectively. Note this command must be used to save the data files 26 and 27, which are required for post-processing or dynamic analysis.

COMMAND>RESE=OFF

This command tells the program not to look at the numbering of the elements nor to resequence them (ie renumber them internally) to make the finite element analysis more efficient. The program is set up to automatically perform this function. Since this model is so small the process is not really necessary. However

when creating a large finite element model this process (ie RESE=ON) is usually necessary because there is definite "pecking order" of how elements and nodes should be numbered. The analysis of a model that does not follow this order will take longer than one that does. This command has the program check the model to see if it conforms to the "pecking order". If the model does not then the program will internally change the element numbering so that it does. The change will not effect the user as it is transparent to the user and the element IDs that the user uses are not changed. For more info refer page 5.3-5 of the NISA II User's Manual

COMMAND>AUTO=ON

This command activates the automatic spurious normal rotation constraint for shells. This input is not specifically required as the default mode is for this to be on. Nodes in basic shell theory have 5 degrees of freedom. When transferring a model from the non-global system to a global system rotation in the theta Z direction may occur. Additionally rotation perpendicular to the theta Z direction may occur. If this happens then the shell (ie model) will not be defined and therefore cannot be solved. Having the AUTO CONSTRAINT on will constrain this perpendicular rotation, thereby allowing the model to be defined.

COMMAND>EIGE=SUBS,CONV

This command tells the program that the eigenvalue extraction method is to be conventional subspace. This command produces some effect on the speed of the analysis. If the eigenvalues are far apart then having the method be accelerated subspace will cause the program to aggressively shift the frequencies in order to speed up the convergence onto the natural frequencies. Refer to pages 3.5-1 to 3.5-4 and 5.3-6 in the NISA II User's Manual.

COMMAND>MASS=LUMPED

This command tells the program to activate the lumped matrix formulation, ie it tells the program to solve the stiffness matrix diagonally instead of consistent with the way the matrix was developed. Refer to page 3.5-5 of the NISA II User's Manual.

COMMAND>*LDCASE,1

This is the load case "control card" for the desired output. Refer to page 7.1-2 of the NISA II User's Manual for an explanation of what the number(s) following the card heading mean. Note when performing an Eigenvalue analysis this card is not applicable. However, using this

card was the only method the author found of "saving" the displacement constraints that were input previously. A more difficult method of inputing the displacement constraints would be to input them directly into the NIS file using a text or line editor, like EDLIIN. Therefore the user should either use this command to save the displacement constraints and then delete the two specific load case lines in the NIS file by using a text or line editor, like EDLIN, or input the displacement constraints directly into the NIS file. (The NIS file will be explained below.) The load case lines that need to be deleted consists of one line containing *LDCASE and a second line containing the string of numbers input immediately after the word *LDCASE above.

COMMAND>*EIGCNTL,2,0,20,0,0.0,0.0,1.0E-5

This command tells the program what the control parameters are for the eigenvalue extraction. Refer to pages 7.1-5 to 7.1-6a of the NISA II User's Manual for an explanation of what each number means.

COMMAND>QUI

Quits the insert mode.

COMMAND>QUI

Quits the DEC mode.

COMMAND>WRI,NISA,EF01.NIS

This takes the above data just input during this session and writes a "NISA" file called "EF01.NIS" that the program uses for it's analysis.

COMMAND>WRI,DBS,EF01.BIN

This takes the above data and writes a binary data base file called "EF01.BIN"

COMMAND>END

This ends the session.

Data Base File: At this time the program will ask if a data base file DISP.BIN is to be written. Since this has already been done with the WRI,DBS,EF01.BIN command above, answer no "N".

The model is now ready to have an eigenvalue analysis performed on it. Once this analysis has been completed a dynamic analysis can be preformed. Remember that files

26.dat and 27.dat must be saved during the eigenvalue analysis in order to be used to complete the dynamic analysis. Refer to the previous section "Program Usage" for guidance on how to perform the analysis.

```
ANAL=FREQUENCY
DAMP=VISCOUS
FILE=EF01
SAVE=26,27
FLOW=0.0
FUPP=10.0
GENF=OFF
INPH=LINEAR
INPO=LINEAR
MRES=OFF
*TITLE
ATTEMPT OF VERIFICATION PROBLEM 8.2
*MODE
1,2,1
*ADDF
0.0,7.5,30
*SPEC
AMPLITUDE SPECTRUM
1001,0,3,1.0,1.0
PHASE SPECTRUM
1002,0,3,1.0,0.0
*DCFO
2,FX,1001,1002,$,200.0
*RSET
1001,2,UX,3,1
*SPOUT
1001,0,1,1001
*ENDDATA
```

ANAL=FREQUENCY

This tells the program that the analysis to be performed is a Frequency Response analysis.

DAMP=VISCOUS

This tells the program that the model has viscous damping in it.

FILE=EF01

This tells program that the prefix EF01 is to be used for the internally generated binary data files. This prefix must be the same as that used for the eigenvalue portion of the problem. The reason for this is because the dynamic analysis must use the 26.dat and 27.dat files generated by the eigenvalue analysis, for the its' analysis. This command provides the prefix name to look for the required data.

SAVE=26,27

This command tells the program to save the internally generated data files, 26.dat and 27.dat. Note this command is not required, as the dynamic program

automatically saves these two files.

FLOW=0.0

This tells the program that the lower limit of the exciting frequencies is 0.0 cycles per time.

FUPP=10.0

This tells the program that the upper limit of the exciting frequencies is 10.0 cycles per time.

Note: In a Frequency Response analysis the program determines the response of the system at various exciting frequencies ranging from the lower limit (FLOW value) to the upper limit (FUPP value).

GENF=OFF

This command tells the program to suppress the generation of frequency points between modes. A system with more than one degree of freedom has more than one natural frequency. The first mode (or fundamental mode) refers to the mode associated with the lowest natural frequency. During a frequency response analysis the program determines the response of the system to frequencies varying between those set with the upper and

lower limit commands. With this command set to off (GENF=OFF) only the response to the natural frequencies occurring between the upper and lower limits will be analyzed. However if this command is on (GENF=ON) then the program will also determine the system's response to various other frequencies equally spaced between the natural frequencies within the upper and lower limits.

INPH=LINEAR

This command tells the program that the effect of any specified phase angle is computed along a linear ordinate and abscissa. The other choices would be to have the ordinate and/or the abscissa be logarithmic.

INPO=LINEAR

This command tells the program that the effect of any specified power spectral density, amplitude spectrum or shock spectrum is computed along a linear ordinate and abscissa. The other choices would be to have the ordinate and/or the abscissa be logarithmic.

MRES=OFF

This tells the program not to (ie to suppress modal response) printout the displacement, velocity and

acceleration responses for all modes. As a result only those modes specified will have responses printed.

From this point on all commands will consists of more than one line of data. As a result a "*" indicates the start of a new set of data.

*TITLE

This is a two line data group. This line tells the program that the following line of input will be the analysis title.

ATTEMPT OF VERIFICATION PROBLEM 8.2

This is the title provided to the analysis which the program will then print at the top of the answer output.

*MODE

This is a multi line data group. This line tells the program that the following line(s) of input will contain the data specifying what modes are to be analyzed. This command is always required, refer to page 8.4-2 of the NISA II User's Manual.

1,2,1

This tells the program that modes 1 and 2, with an increment of 1 between them, are to be analyzed with zero "0" damping. This is accomplished with the following format: Entry 1,2,3,4 where:

Entry 1 is the first mode to be analyzed.

Entry 2 is the last mode to be analyzed.

Entry 3 is the increment between modes to be analyzed. Suppose the modes to be analyzed were modes 1, 5 and 10 then entry 1 would be 1, entry 2 would be 10 and entry 3 would be 2.

Entry 4 is the damping identification # that corresponds to the damping parameters set previously. The damping ID # allows a system to be analyzed for various damping values without remodeling the whole system each time. In this example there is zero "0" damping. Therefore, there is no damping data input. However, suppose it was desired to analyze this problem with two different damping values, say 30% and 70% of critical. To do this two damping data groups would be required. Part of the damping data input is an arbitrary ID #. Let's let the 30% damping have an ID # of 1030 and the 70% have an ID # of 170 (note these numbers are totally arbitrary). Now to find the response of this system with those two values input:

*MODE

1,2,1,1030

1,2,1,170

*ADDF

This is a two line data group. This line tells the program that the following line of input will contain the data specifying the exciting frequencies that are to be added to the system.

0.0,7.5,30

This tells the program that the system is to be analyzed for 30 exciting frequencies equally spaced between 0.0 cycles per time and 7.5 cycles per time. This is accomplished with the following format: Entry 1,2,3 where:

Entry 1 is the value of the starting frequency.

Entry 2 is the value of the ending frequency.

Entry 3 is the number of equal increments between entry 1 and 2 that are to be analyzed.

*SPEC

This is a multi line data group. This line tells the program that the following lines of input will provide the

spectrum definition. This is accomplished with multiple two line series where the first line provides the spectrum description and the second line provides the data.

AMPLITUDE SPECTRUM

This tells the program that the following line of data defines the amplitude spectrum.

1001,0,3,1.0,1.0

This tells the program that the amplitude spectrum has a constant value of 1.0, a scale factor of 1.0 and an ID # of 1001. This is accomplished with the following format: Entry 1,2,3,4,5 where:

Entry 1 is an arbitrarily assigned #.

Entry 2 is the number of points used to define the spectrum. If the spectrum is constant this entry is ignored. If the spectrum is not constant this entry must be greater than or equal to two.

Entry 3 is a number between 0 and 3 which is used to define the input spectrum. "0" - spectrum is a function of frequency (cycles/time); "1" - spectrum is a function of frequency (radians/time); "2" - spectrum is a function of period; "3" - spectrum is constant.

Entry 4 is the scale factor for the spectrum data.

Entry 5 is the constant spectrum value.

Note if the spectrum is not constant then the next line(s) of input would define the spectrum. This would be accomplished with each line containing a maximum of three pairs of numbers. The first number of the pair is the frequency/period value. The second number is the spectrum value that corresponds with the frequency/period value. The total number of pairs must equal the value placed in entry 2 above.

PHASE SPECTRUM

This tells the program that the following line of data defines the phase spectrum.

1002,0,3,1.0,0.0

This tells the program that the phase spectrum has a constant value of 0.0, a scale factor of 1.0 and has an ID # of 1002. Refer to the above description for format and entry discussions.

For every line of Amplitude Spectrum data there has to be a corresponding line of Phase Spectrum data. The first entry of the data lines must be different. For example

suppose there are two dynamic forces being applied to a model. One of the forces is out of phase with the other force by 90 degrees. This would be modeled as follows:

```
Amplitude Spectrum
1001,0,3,1,1
Amplitude Spectrum
1002,0,3,1,1
Phase Spectrum
1003,0,3,1,0
Phase Spectrum
1004,0,3,1,1.571
```

The description "Amplitude Spectrum" or "Phase Spectrum" must proceed every line of data. Also the Phase Spectrum data line 1003 corresponds to Amplitude Spectrum data line 1001 and 1004 corresponds with 1002. Refer to the above description for an explanation of what each line of data means.

*DCFO

This is a multi line data group. This line tells the program that the following line(s) of input will contain the data used to specify a concentrated nodal force.

2,FX,1001,1002,$,200.0

This tells the program there is a concentrated force of 200.0 units in the X direction at node 2 for both the

amplitude and phase spectrums. This is accomplished with the following format: Entry 1,2,3,4,5,6,$,7 where:

Entry 1 is the node number the force or moment is applied to.

Entry 2 is the force or moment direction.

Entries 3 & 4 are the ID #s that corresponds to the amplitude and phase spectrums.

Entry 5 is the last node that the force or moment is applied to.

Entry 6 is the increment between the first and last nodes this force or moment will be applied to.

$ This tells the program to skip all entries between the last one inputed and entry 7.

Entry 7 is the value of the force multiplier. Basically there are two numbers that can be input here. The correct number depends on the spectrum definition. The easiest way to explain this is with an example. Therefore, let the forcing function = 200 sine 400 theta. Now if the amplitude definition was based on 200 sine 400 theta (ie theta = 0.5236 radians and the force = 173.2) then the force multiplier is 1. However if the amplitude was based on sine 400 theta (ie theta = 0.5236 radians and the force = 0.866) then the force multiplier = 200. However, any number can be used as the force

multiplier, as long as the user maintains consistency through out the modal.

*RSET

This is a multi line data group. This line tells the program that the following line(s) of input will define the nodal/element response quantities that are to be provided by the program. This command is required and there must be a minimum of one line of *RSET data for each line of AMPLITUDE SPECTRUM. Refer to page 8.7-2 of the NISA II User's Manual.

1001,2,UX,3,1

This tells the program that the response at nodes 2 and 3, in the X direction, is to be provided with a response ID # of 1001. This is accomplished with the following format: Entry 1,2,3,4,5,$,6 where:

Entry 1 is an arbitrarily assigned response ID #.

Entry 2 is the node or element # to be analyzed.

Entry 3 is the response to be specifically provided.

Entry 4 is the last node or element # to be analyzed.

Entry 5 is the node or element increment

(between the entry 2 node/element and entry 4 node/element).

$ This tells the program to skip all entries between the last one inputed and entry 6.

Entry 6 is the layer number for composite elements.

Note: Additional input lines can be used to specify responses for different nodes/elements and/or different responses. For example suppose the responses at node 2 in both the X and Y directions was desired. Then there would be two lines of input, one with entry 3 containing UX and one with entry 3 containing UY.

*SPOUT

This is a multi line data group. This line tells the program that following line(s) of data will provide the required output responses.

1001,0,1,1001

This tells the program that the output spectr, response maxima and, the frequencies at which they occur are to be printed, with a zero "0" phase angle shift, for response ID set 1001. This output will have an identification number 1001, which is referenced during post processing. This is accomplished with the following

format: Entry 1,2,3,$,4-13 where:

Entry 1 is the output identification # to be referenced during post processing.

Entry 2 is the phase angle shift.

Entry 3 is the printout option: "0" = output spectra is calculated but only the response maxima and the frequencies at which they occur are printed; "1" = output spectra, response maxima and the frequencies at which they occur are printed. "2" = the response is printed at the specified phase angle shift.

$ tells the program to skip all entries between the last one inputed and entry 4.

Entry 4 to 13 are the response set identification number(s) (up to ten) for which the output is required. This (these) number(s) correspond to the response output listed in the RSET command.

*ENDDATA

This tells the program that this is the last line of data.

Once this file is completed and the eigenvalue analysis has been performed the model is ready to have the dynamic analysis performed. To perform the dynamic analysis

access the directory and sub-directory that the above file is saved in, type "NISA286" and hit the return. Now move the cursor, using the arrow keys, to the "DYNAMIC" selection and hit the return. At this time the program will ask for the input file name. Type in the name of the file above and hit the return. Now the program will ask for the output file name. Type in the desired output name and hit return. Now wait for the analysis.

CHAPTER THREE

PROGRAM UTILIZATION AND COMPARISON

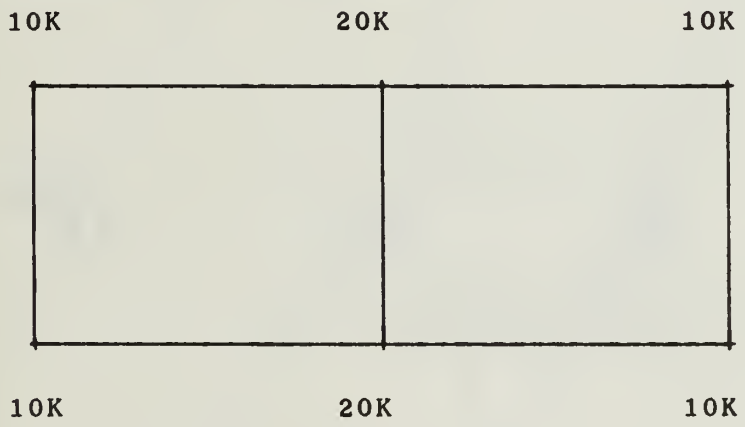
INTRODUCTION

Chapter Two provided a user's manual for the NISA II program as well as a detailed step by step explanation of how to generate a finite element model and analysis it for Static, Eigenvalue, Transient Dynamic, and Frequency Response conditions. This "user's manual" will now be utilized to model and analyze two different problems, a rigid frame on a slab on springs and a circular foundation on a halfspace.

FRAME WITH SLAB ON SPRINGS

The following example consists of a Rigid frame on a concrete slab on springs. This example was generated using DISPLAY II and then analyzed for both a static response and a frequency response using the NISA II program. This analysis is presented in the following order: Model sketch, DISPLAY II input, input explanation, frequency response input, input explanation, and response output. A GTSTRUDL response is currently being analyzed and will be placed into the Appendix D of this report, for a comparison with NISA II, when it is completed.

This example consists of a rigid frame on a concrete slab, on springs. Machinery located in the center of the rigid frame causes a sinusoidal dynamic force of 60,000 # in the X direction and 30,000 # in the Z direction. These two forces are evenly distributed between two points, such that Fx=30,000 # and Fz=15,000 # at each point, are occurring at a frequency of 400 rpm, and Fz is 90 degrees out of phase with Fx. The weight of the machinery is distributed at the top of the rigid frame as follows:



Material properties are as follows:

columns (c1)= W18x119 beams (b1)= W36x230

concrete slab 3ft thick (b2)= W30x173

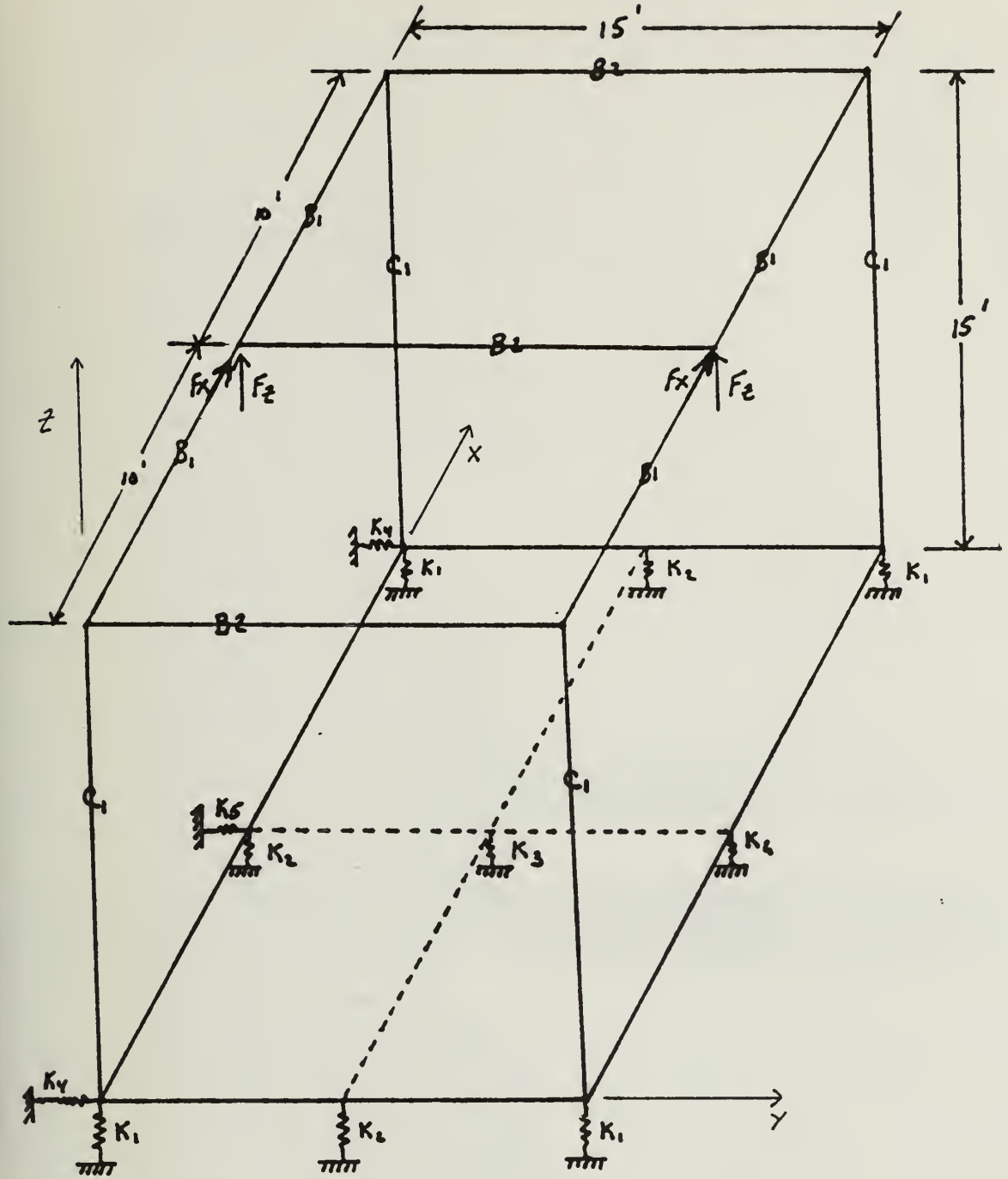
poissons ratio = 0.17

E = 3.6E6 psi

gamma = 150 pcf

Springs (kips per inch)

K1 = 586 K2 = 1170 K3 = 2340 K4 = 2500 K5 = 5000


15'
B2
10'
B1
C1
B1
C1
15'
B2
Fx
Fz
Fx
Fz
Z
X
10'
B1
B1
Ky
K1
K2
K1
B2
C1
C1
K5
K2
K3
K4
Ky
K1
K2
K1
Y

```
GRD,ADD,1,0/0/0
GRD,TRS,1,2,240
LIN,2GD,1,2,1
LIN,TRS,1,2,/180
PAT,2LN,1,2,1
VEW,ABS,-80/0/10
PLO
PAT,TRS,1,2,//-20
FEG,QUA,1,,,1/2/2/20,1/1
FEG,QUA,2,,,1/2/2/20,1/1
ELE,DEL,5T8
NOD,ADD,19,/-20
NOD,ADD,20,120/-20
NOD,ADD,21,240/-20
ELE,ADD,5,17/1//2,1/10
ELE,ADD,6,17/1//3,2/11
ELE,ADD,7,17/1//2,3/12
ELE,ADD,8,17/1//3,4/13
ELE,ADD,9,17/1//4,5/14
ELE,ADD,10,17/1//3,6/15
ELE,ADD,11,17/1//2,7/16
ELE,ADD,12,17/1//3,8/17
ELE,ADD,13,17/1//2,9/18
ELE,ADD,14,17/1//5,1/19
ELE,ADD,15,17/1//6,2/20
ELE,ADD,16,17/1//5,3/21
NOD,ADD,22,//180
NOD,ADD,23,120//180
NOD,ADD,24,240//180
NOD,ADD,25,/180/180
NOD,ADD,26,120/180/180
NOD,ADD,27,240/180/180
ELE,ADD,17,12/1/2/7,1/22
ELE,ADD,18,12/1/2/7,3/24
ELE,ADD,19,12/1/2/7,7/25
ELE,ADD,20,12/1/2/7,9/27
ELE,ADD,21,12/1/2/8,22/23
ELE,ADD,22,12/1/2/8,23/24
ELE,ADD,23,12/1/2/8,25/26
ELE,ADD,24,12/1/2/8,26/27
ELE,ADD,25,12/1/2/8,22/25
ELE,ADD,26,12/1/2/8,23/26
ELE,ADD,27,12/1/2/8,24/27
ELE,ADD,28,25/1//10,22
ELE,ADD,29,25/1//11,23
ELE,ADD,30,25/1//10,24
ELE,ADD,31,25/1//10,25
ELE,ADD,32,25/1//11,26
ELE,ADD,33,25/1//10,27
PRO,ADD,1,36/36/36/36
```

```
MAT,ADD,1,3.6E6/0.1/////////2.246E-4
PRO,ADD,2,5.85E5
PRO,ADD,3,1.17E6
PRO,ADD,4,2.34E6
PRO,ADD,5,2.5E6
PRO,ADD,6,5.0E6
PRO,ADD,7,35.1/253/2190
PRO,ADD,8,67.6/940/15000
PRO,ADD,9,50.8/598/8200
MAT,ADD,2,2.9E7/0.3////////7.332E-4
PRO,ADD,10,25.88
PRO,ADD,11,51.76
DISP,ADD,1T3,1,0///0/0/0
DISP,ADD,4T9,1,0/0//0/0/0
DISP,ADD,10T21,1,0/0/0/0/0/0
DISP,ADD,22T27,1.///0/0/0
FOR,ADD,1,1,//-386.4
DEC
INS
ANAL=STATIC
FILE=GTSTAT
SAVE=26,27
*TITLE,ANALYSIS OF CE6193 GTSTRUDL EXAMPLE
*LDCASE,1
*EIGCNTL,1,0,20,0,0.0,0.0,1.0E-5
*MODEO,1,20,1,1
*EIGO,1,1,0,3,1,0,1
QUI
QUI
WRI,NISA,GTSTAT.NIS
WRI,NISA,GTEIGE.NIS
WRI,DBS,GTSTAT.BIN
WRI,DBS,GTEIGE.BIN
END
N
```

The two previous pages contain the commands required to generate this model using the DISPLAY II program. With the exception of the following clarifications/explanations all commands are explained in Chapter Two.

COMMAND>VEW,ABS,-80/0/10

This command tells the program to change the angle the user is viewing the model with. This actual command will rotate the model clockwise 80 degrees about the X axis and 10 degrees counterclockwise about the Z axis (counterclockwise angles are positive). This "new" viewing angle will now be used for all future model "plots".

COMMAND>PLO

This command tells the program to erase the current screen picture and to replot the model. The previous command changed the viewing angle and this new "plot" will be with the new viewing angle.

COMMAND>FOR,ADD,1,1,//-386.4

The explanations for this command provided in Chapter Two indicate that this command tells the program that there is a force on 386.4 units in the -Z direction on node #1. However upon examining the figure on page 3.4 it

is noted that node #1 has no such force. Therefore why is this command added? It was discovered while developing this model that if a static analysis is to be performed considering just the mass of the model then the command "BODYFORCE" must be used. However the DISPLAY II program does not accept this command. As a result the author used the command "FOR,ADD,..." for two purposes:

1. As a reminder to add the "BODYFORCE" command to the ".NIS" file created by the DISPLAY II program.
2. As an easy method of determining the correct location of inserting the "BODYFORCE" command to the ".NIS" file. This is accomplished by changing the "FORCE" data lines in the ".NIS" file to "BODYFORCE" data lines using a line or text editor. Refer to page 7.3-11 of the NISA II User's Manual for the correct "BODYFORCE" format.

COMMAND>*MODEO,1,20,1,1

This command and the "*EIGO" command were both discussed in Chapter Two. However this is a reminder that the DISPLAY II program does not accept these commands and that they must be inserted into the ".NIS" file using a line or text editor.

COMMAND>WRI,...

All the examples discussed in Chapter Two had one "WRI,NISA,..." and one "WRI,DBS,..." command. This model has two of each. This may not be required; however, since the .NIS file for a static analysis contains a few different commands than the .NIS file for an eigenvalue analysis, the author felt that the easiest way to perform both analyses was to let the DISPLAY II program create two .NIS files, one for the static analysis and one for the eigenvalue analysis. Once the two files are created, each may then be easily tailored using a text or line editor, for the specific analysis.

```
ANAL=FREQUENCY
DAMP=VISCOUS
FILE=GTEIGE
GENF=OFF
*TITLE
FREQUENCY RESPONSE OF CE6193 GTSTRUDL EXAMPLE
*MODE
1,1,1,1001
*DAMP
1001,.1
*ADDF
41.888
*SPEC
AMPLITUDE SPECTRUM
1001,0,3,1.0,1.0
AMPLITUDE SPECTRUM
1002,0,3,1,1
PHASE SPECTRUM
1003,0,3,1.0,0
PHASE SPECTRUM
1004,0,3,1,1.571
*DCFO
23,FX,1001,1003,$,30000.0
26,FX,1001,1003,$,30000.0
23,FZ,1002,1004,$,15000.0
26,FZ,1002,1004,$,15000.0
*RSET
1001,1,UZ,9,1
1001,22,UZ,27,1
1002,1,UZ,9,1
1002,22,UZ,27,1
*SPOUT
1001,0,0,1001
1002,0,0,1002
*ENDDATA
```

FREQUENCY RESPONSE INPUT EXPLANATION

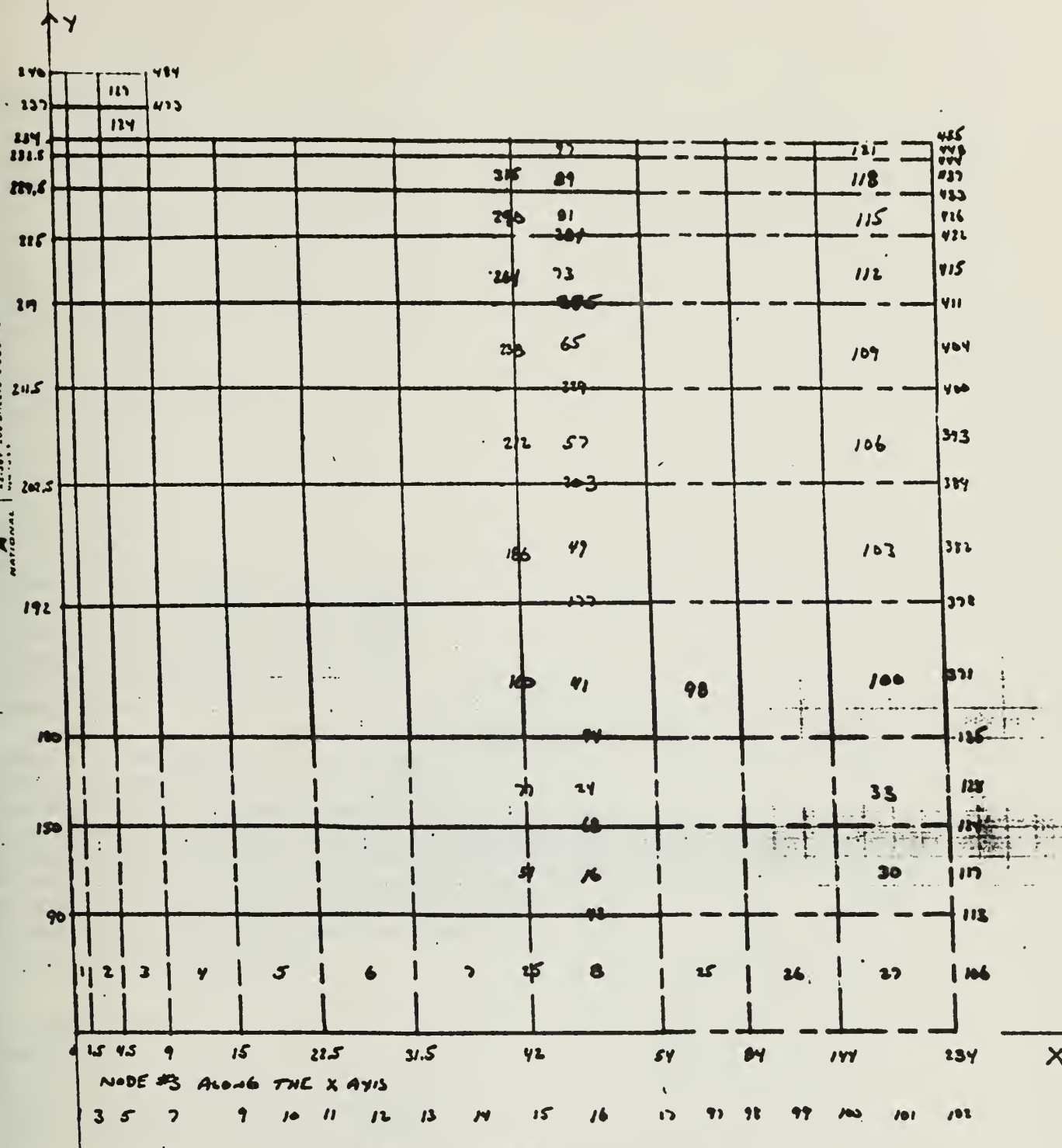
All the commands listed in the frequency response input have been explained in Chapter Two. Accordingly a line by line description is not provided. However the following general description is provided.

This model consists of a rigid frame, on a concrete slab, on springs. There are two forces being applied to nodes 23 and 26, one in the X direction and one in the Z direction. The Z direction force is 90 degrees out of phase with the X direction force. An analysis of the displacement is the Z direction for nodes 1 to 9 and 22 to 27 is required.

CIRCULAR FOUNDATION ON A HALFSPACE

The following example consists of a circular foundation on a halfspace. A halfspace has no boundaries along three edges, typically along the X directions and one of either the Y or Z directions. Since an infinite length cannot really be modeled, a halfspace is modeled by making the halfspace boundaries many times larger than the diameter of the foundation on the halfspace. In this model the foundation has a diameter of 18 feet and the boundaries are located 234 feet from the foundation's center. Additionally the use of symmetry is used in this model. As a result only one half of the model is analyzed. This example was generated using DISPLAY II and then analyzed for a frequency response using the NISA II program. The NISA II analysis was then compared to theoretical analyses of the same model. This analysis is presented in the following order: Model sketch, DISPLAY II input, input explanation, frequency response input, input explanation.

Appendix E contains the analyses comparisons, hand calculations, graphs, and computer printouts used for the theoretical analyses.



THIS IS A 127 ELEMENT MESH; 9 FT RADIAS BY 6FT THICK FOUNDATION ON THE SURFACE. THE SOIL IS REPRESENTED BY 121 ELEMENTS (234' X 234'; 11 ELEMENTS X 11 ELEMENTS). ELEMENTS CONTAIN 8 NODES EACH. THE NUMBERS ALONG THE AXES REPRESENT THE DISTANCE FROM THE ORIGIN THE SOLID ELEMENTS DRAWN APPROXIMATELY TO SCALE (1" = 135'). THE DASHED ELEMENTS ARE NOT TO SCALE. ALL NODES ALONG THE OUTSIDE VERTICAL EDGES ARE PROHIBITED FROM MOVEMENT IN THE X DIRECTION. ALL NODES ALONG THE BOTTOM HORIZONTAL EDGE ARE PROHIBITED FROM MOVEMENT IN BOTH THE X & Y DIRECTIONS.

```
GRD,ADD,1,0/0/0
GRD,TRS,1,2,54
GRD,TRS,1,3,234
LIN,2GD,1,2,1
LIN,2GD,2,3,2
LIN,TRS,1,3,/180
LIN,2GD,1,5,/234
LIN,DEL 2GD
LIN,DEL,4
LIN,TRS,1,5,/234
LIN,TRS,2,4,/180
LIN,TRS,2,6,/234
GRD,ADD,10,9/234
LIN,2GD,6,10,7
LIN,TRS,7,8,/6
PAT,2LN,1,3,1
PAT,2LN,2,4,2
PAT,2LN,3,5,3
PAT,2LN,4,6,4
PAT,2LN,7,8,5
FEG,QUA,1,,,2/8/3/3,1/1/8/0.3333
FEG,QUA,2,,,2/3/3/3,1/1/3/0.3333
FEG,QUA,3,,,2/8/8/3,1/1/8/0.125
FEG,QUA,4,,,2/3/8/3,1/1/3/0.125
FEG,QUA,5,,,2/3/2/3,2/2/3/1
NOD,MER,1T484
N
MAT,ADD,1,6.0E5/0.45////////3.88
MAT,ADD,2,5.76E8/0.45////////4.66
DISP,ADD,478/474/467/463/344/335/318/309,1,0
DISP,ADD,292/283/266/257/240/231/214/205,1,0
DISP,ADD,188/179/162/153/79/70/53/44,1,0
DISP,ADD,27/18/106/113/117/124/128/135,1,0
DISP,ADD,371/378/382/389/393/400/404/411,1,0
DISP,ADD,415/422/426/433/437/444/448/455,1,0
DISP,ADD,1T17/97T102,1,0
DEC
INS
ANAL=EIGENVALUE
FILE=SOLID
SAVE=26,27
*TITLE,CIRCULAR FDN ON A HALFSPACE
*EIGCN,40,0,40,0,0.0,0.0,1.0E-5
*LDCASE,1
QUI
QUI
WRI,NISA,SOLID.NIS
WRI,DBS,SOLID.BIN
END
N
```

The previous page contains the commands required to generate this model using the DISPLAY II program. With the exception of the following clarifications/explanations all commands are explained in Chapter Two.

COMMAND>NOD,MER,1T484

This command tells the program to review all nodes between 1 and 484 (all the nodes of the model) and to combine any two or more nodes together, into one, that are located within the "default" distance of each other. This is accomplished with the following format: "NOD,MER,3,4" where:

Entry 3 is the list of nodes to be merged.

Entry 4 is the "tolerance", ie. the specified distance between nodes.

If a model was generated with more than one patch this command must be used to "tie" the patches together. If this command is not used on a model created by more than one patch, it cannot be analyzed. Refer to pages 4.3-12 to 4.3-14 of the DISPLAY II User's Manual for more information.

COMMAND>N

If no tolerance is specified in the above command the program will ask if the user would like to modify the

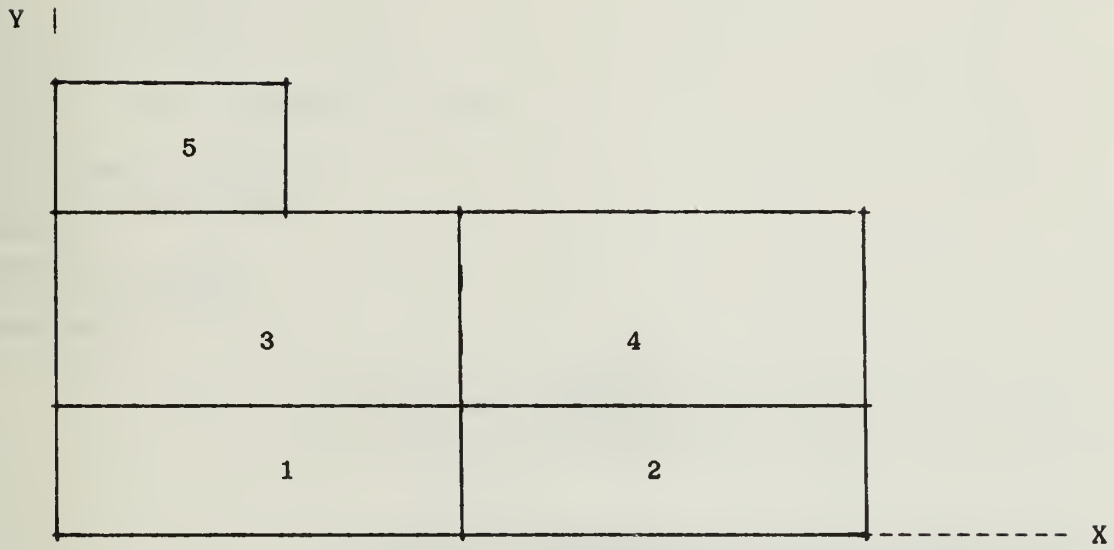
default tolerance (Y/N)? The author chose to use the default tolerance for this merge and therefore responded with "N".

The element type used in this model and analysis was NKTP 3, an axisymmetric solid element. This was the first analysis the author performed using this type of element as well as the first model generated using multiple patches. While performing this model generation and analysis the following restrictions, not found in any of the NISA II User's Manuals, were discovered:

The NISA II User's Manual states that "The element must lie in the right half of the global XY plane where the global X-axis is the radial direction, and the global Y-axis is the axis of revolution or axis of symmetry." The author determined that not only does this element have to remain in the right half of the global XY plane but it must also remain in the first quadrant of the global coordinate system, ie all nodal coordinates must be positive.

All the element meshes must be generated starting with the patch closest to the origin of the coordinate system, move left along the X-axis, then move up to the next layer of patches and move left along the X-axis from the Y-axis again. The following figure contains 5 patches, blocks 1 to 5. To correctly generate a finite element mesh from these patches, start with patch 1 and

finish with patch 5, completing the generation in numerical order.



```
ANAL=FREQUENCY
DAMP=VISCOUS
FILE=SOLID
GENF=OFF
*TITLE
SINUSOIDAL LOADING ON A CIRCULAR FDN ON A HALFSPACE
*MODE
1,39,1,1001
*DAMP
1001,0
*ADDF
4.0,8.0,1
*SPEC
AMPLITUDE SPECTRUM
1001,0,3,1.0,1.0
PHASE SPECTRUM
1002,0,3,1.0,0.0
*DCFO
478,FY,1001,1002,478,1,50000
*RSET
1001,344,UY,360,1
1002,344,UX,360,1
*SPOUT
1001,0,0,1001
1002,0,0,1002
*ENDDATA
```

FREQUENCY RESPONSE INPUT OF A CIRCULAR FOUNDATION ON A HALFSPACE

All the commands listed in the frequency response input, shown on the previous page, have been explained in Chapter Two. Accordingly a line by line description is not provided. However the following general description is provided.

This model consists of an 18 foot diameter circular foundation on a halfspace. The halfspace is modeled by setting a displacement constraint in the X and Y direction 234 feet below the base of the foundation and setting a displacement constraint in the X direction along the center of the foundation and 234 feet to the right of the foundation's center. There is one force being applied to the center of the foundation, node 478, in the Y direction. An analysis of the displacements in the X and Y directions for nodes 244 to 360 is required. These nodes are located on the surface (top) of the halfspace starting at the center of the foundation and ending 54 feet away.

CHAPTER FOUR

CONCLUSIONS

The following conclusions can be made based on the results of this research problem:

1. There are many different Finite Element Analysis programs available on the market today. One of the programs available for use on a personal computer is called NISA II and is produced by Engineering Mechanics Research Corporation located in Troy Michigan. NISA II is a very powerful program that is relatively user friendly, once the user becomes familiar with it, provides rapid analysis, and is fairly inexpensive to use on a personal computer. Additionally it provides answers in good agreement with check cases.

2. Even though a program is considered user friendly, as in the case of the NISA II program, it still requires an extensive amount of "user time" for someone who is unfamiliar with the program to become proficient in its utilization. A large amount of this user time is typically spent trying to solve/understand roadblocks encountered that are not clearly explained in the program provided user manuals. Once an individual becomes proficient with a program, "reasonably" correct answers should be provided.

3. The NISA II User's Manuals provide a broad explanation on how to use the program. This is fine for

someone who is familiar with the basics of a finite element modeling program. This report, on the other hand, is designed to provide a much more basic user's manual on how to use the NISA II computer program. As a result this report gives complete step by step examples on how to develop and analyze various types of static and dynamic problems, as well as invaluable hints on where to avoid problems not discussed in the NISA II User's Manuals.

4. The comparison between NISA and other published values, of an axisymmetric solid on a halfspace under a sinusoidal loading, show the following:

A. All values, even those of the published solutions, tend to have rather large differences.

B. NISA II values tend to compare reasonable well, not varying any greater than any of the other solutions.

C. The NISA II values were for a "model" with rigid boundaries. These boundaries were located so far away from the loaded solid, that it was presumed they would have no effect. However as no other finite element solution was obtained, for a comparison, this can not be completely determined.

5. When comparing the NISA II solutions to those calculated using Lysmer it was noted that Lysmer provided displacements in excess of six times the amount provided

by NISA II. The following are possible reasons:

A. The NISA II model had rigid boundaries, which may have allowed energy to bounce off, where as the Lysmer finite element model had energy adsorbing boundaries.

B. The Lysmer curve of frequency verses displacement had only one peak. The NISA II curve of frequency verses displacement gave multiple peaks. The reason for the multiple peaks could be due to the rigid boundaries and therefore, this needs to be looked into more.

C. As a result of the above it is highly recommended that an exact comparison between NISA and another Finite Element program be accomplished. Based on discussions between the author and the author's advisor it is understood that this comparison will be done in the near future.

CHAPTER FIVE

BIBLIOGRAPHY

BIBLIOGRAPHY

1. Gazetas, G., "Summary of Analysis of Machine Foundation Vibrations: State-of-the-Art". Soil Dynamics and Earthquake Engineering, Vol. 2, No. 1, 1983.

2. Luco, J.E., and Westmann, R.A., "Dynamic Response of Circular Footings", UCLA-ENG-7113, April 1971.

3. Lysmer, Report by Lysmer, University of California, Berkley, from CE-6193 Dynamics of Massive Media class notes.

4. NAVFAC, "Soil Dynamics, Deep Stabilization, and Special Geotechnical Construction", NAVFAC DM-7.3

5. Engineering Mechanics Research Corporation, NISA II User's Manual, EMRC Troy Michigan 48099.

6. Engineering Mechanics Research Corporation, DISPLAY II User's Manual, EMRC Troy Michigan 48099

7. Engineering Mechanics Research Corporation, Verification Problems Manual, EMRC Troy Michigan 48099

APPENDIX A

REQUIRED EQUIPMENT

REQUIRED EQUIPMENT

In order to use the NISA II program, version 88.7, on a PC the following minimum equipment is required:

1. IBM PC-XT or IBM PC-AT compatible Personal Computer. The author used a HP 386.
2. 640K bytes of central memory.
3. IBM compatible Color Graphics Adaptor (CGA) and monitor.
4. A 20 MB hard disk with a 360 KB floppy drive.
5. A Math Coprocessor (Intel 8087 of 80287 chip).
6. A 132 character line printer.
7. MS-DOS Version 2.1 of above operating system.
8. A Parallel port for the protection device supplied by EMRC.

Additionally the following equipment is Optional:

1. IBM compatible Enhanced Graphics Adaptor (EGA) and Enhanced Display monitor.
2. IBM compatible Professional Graphics Controller (PGA) and Professional Display monitor.
3. Ink jet printers (ie Tektronix 4696) and screen dump software for EGA and PGA.

The author encountered a problem during the initial set up

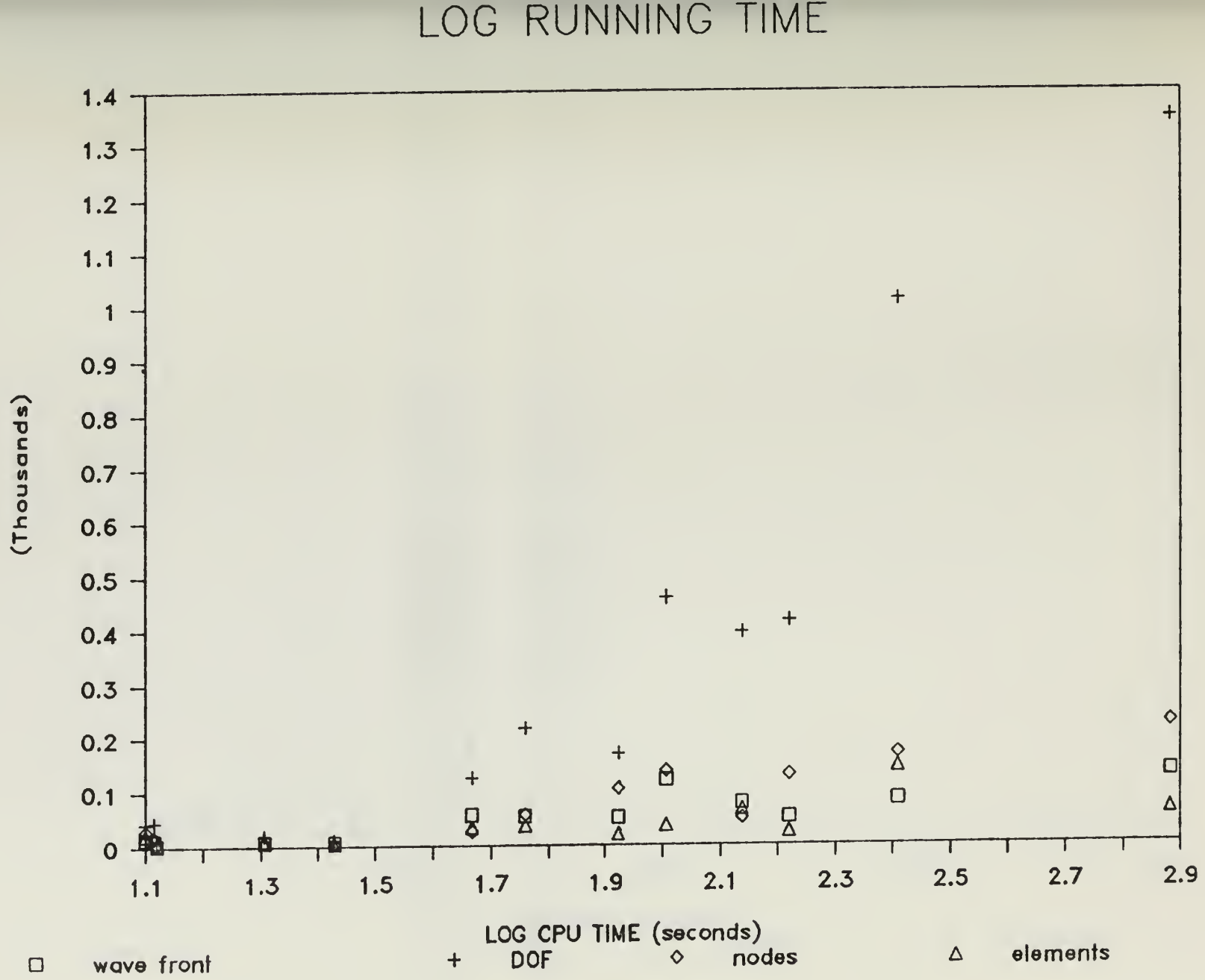
and operation of the program. This problem was due to the MS-DOS version (DOS 3.3) and all the associated software requiring too much space. As a result the hardware did not have 600K bytes of RAM available for the NISA II and DISPLAY II programs. To over come this problem the author deleted everything except the main DOS 3.3 from the operating systems start up path. This resulted in having about 596K of RAM available for the NISA II and DISPLAY II programs to operate, which was enough.

APPENDIX B

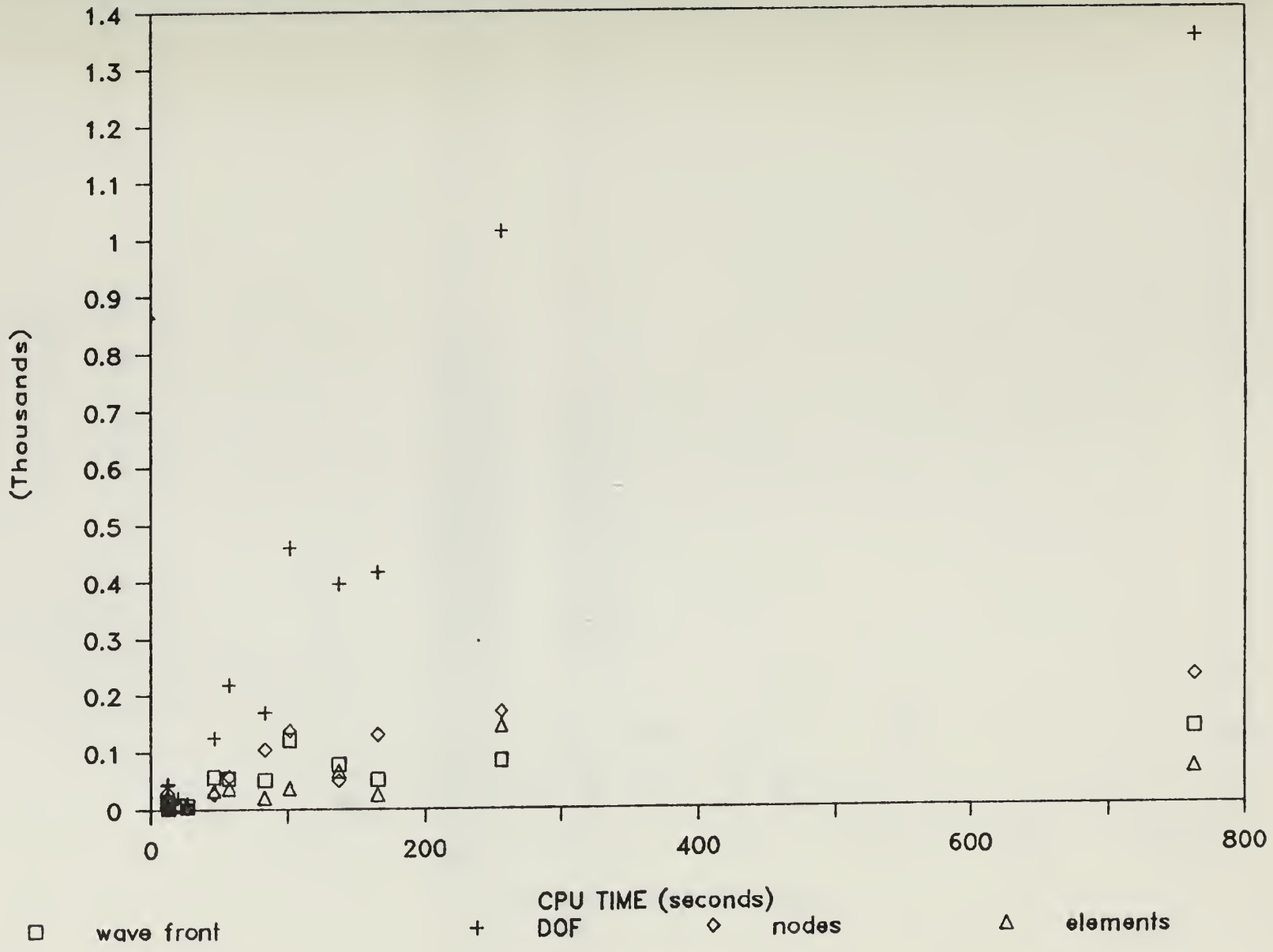
RUNNING TIME

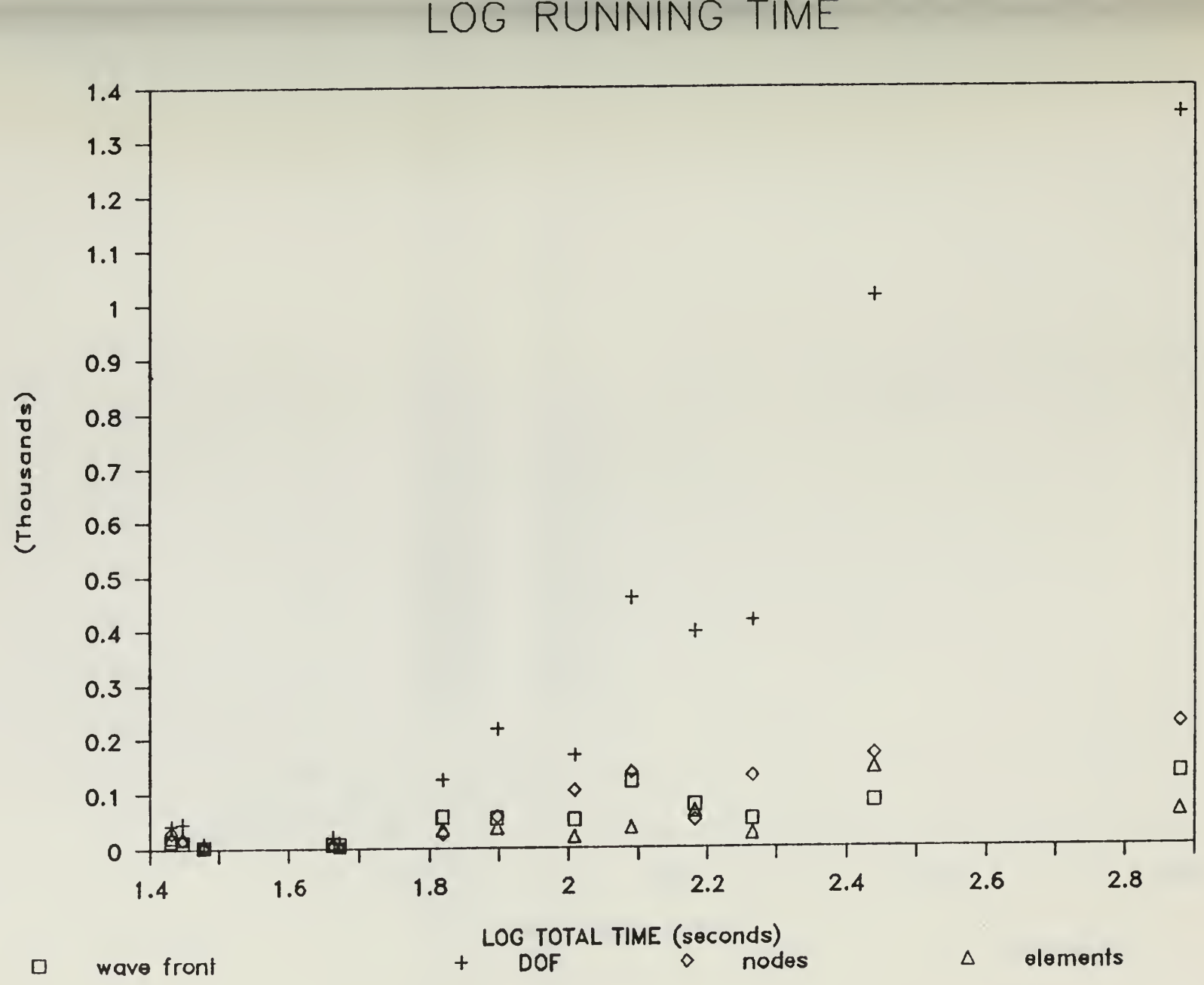
The author used a HP VECTRA QS/16 to perform all analysis on. The actual time required to perform an analysis depends more on the type of problem being analyzed than anything else. However in an attempt to provide some type of guide to running time the following graphs are provided. As shown on these graphs, whether it is the log total time or just cpu time, the number of degrees of freedom is the controlling factor on length of time required to perform an analysis.

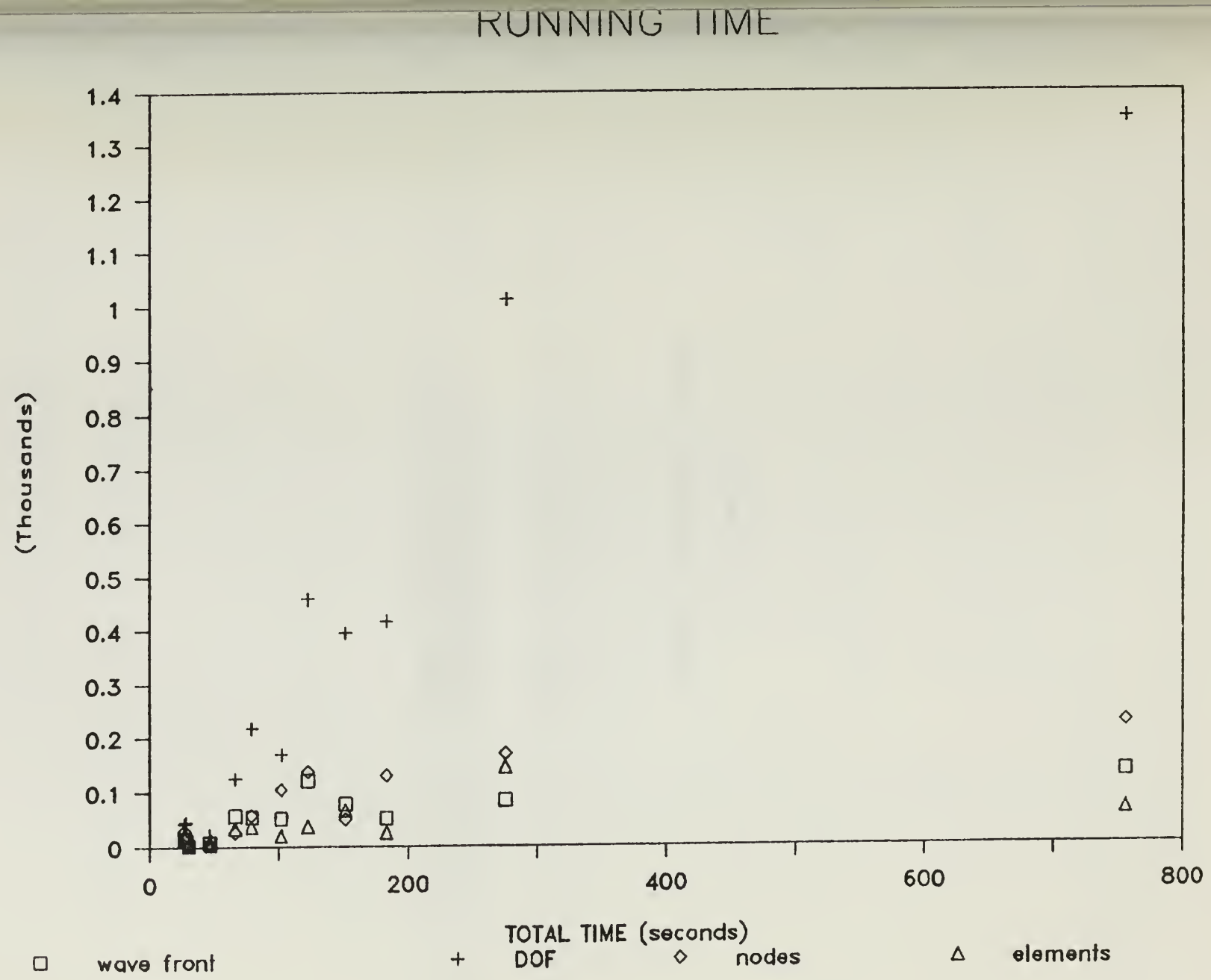
In addition to the above "guide", the following is provided. Two eigenvalue analyses were performed on a model with: 860 DOF, 430 nodes, 127 elements, and a wave front of 84. One analysis used "accelerated subspace" eigenvalue extraction and the other analysis used "inverse" eigenvalue extraction. The accelerated subspace method took approximately 3993 seconds (about 67 minutes) to provide the first 40 eigenvalues. The inverse method took approximately 7708 seconds (about 1 1/2 hours) to extract 30 eigenvalues starting with a frequency of 2 cps.


LOG RUNNING TIME
1.4
1.3
1.2
1.1
1
0.9
0.8
0.7
0.6
0.5
0.4
0.3
0.2
0.1
0
(Thousands)
1.1
1.3
1.5
1.7
1.9
2.1
2.3
2.5
2.7
2.9
LOG CPU TIME (seconds)
wave front
DOF
nodes
elements

RUNNING TIME




LOG RUNNING TIME
(Thousands)
1.4
1.3
1.2
1.1
1
0.9
0.8
0.7
0.6
0.5
0.4
0.3
0.2
0.1
0
1.4
1.6
1.8
2
2.2
2.4
2.6
2.8
LOG TOTAL TIME (seconds)
wave front
DOF
nodes
elements


RUNNING TIME
1.4
1.3
1.2
1.1
1
0.9
0.8
0.7
0.6
0.5
0.4
0.3
0.2
0.1
0
(Thousands)
0
200
400
600
800
TOTAL TIME (seconds)
wave front
DOF
nodes
elements

APPENDIX C

NISA II EXECUTIVE COMMANDS

Table 5.2.3 Alphabetical list of *all* executive commands for *all* analyses

Executive Command Name[1]	Description	Applicable Analysis Types[2]
ACUTOFF	Range of model coupling	RA
ANALYSIS	Specify analysis type	ALL1, ALL2
AUTO CONSTRAINT	Spurious normal rotation constraint	ST, EV, BU
BLANK COMMON	Blank common storage unit	ALL1, ALL2
DAMPING	Type of damping	ALL2
DELT TIME	Time increment	TR
DIRECTION	Directional superposition procedure	SHO
EIGEN EXTRACTION	Eigenvalue extraction method	EV, BU
ELEMENT ECHO	Echo element input	ALL1
EXECUTION	Select execution/checking run	ALL1, ALL2
ENDTIME	Ending time	TR
FILENAME	Specify NISA file prefix	ALL1, ALL2
FLOWER	Lower limit of exciting frequency	RA, FR
FRQRDF	Reduction factor for exciting frequency point generation	RA, FR
FSMALL	Frequency cut-off to identify rigid body modes	ALL2
FUPPER	Upper limit of exciting frequency	RA, FA
GENFREQUENCY	Control generation of exciting frequency points	RA, FA
GFACTOR	Acceleration due to gravity	ALL2
INITIAL TEMPERATURE	Define initial temperature	TH
INPOLATION	Type of interpolation	RA, FR, SHO
INPHASE	Type of interpolation for phase spectra	FR
INTEGRATION	Type of integration procedure	RA
MASS FORMULATION	Mass formulation for eigenvalue analysis	EV

(1) Minimum abbreviations are in bold face

(2) ST: Linear static
EV: Eigenvalue
BU: Buckling
NL: Nonlinear static
SH: Steady state heat transfer
TH: Transient heat transfer
TR: Transient dynamics
FR: Frequency response
RA: Random vibration
SHO: Shock spectrum
ALL1: All static and heat transfer analyses (ST, EV, BU, NL, SH, TH)
ALL2: All modal dynamic analyses (TR, FR, RA, SHO)

Table 5.2.3 Alphabetical list of *all* executive commands for *all* analyses (continued)

Executive Command Name[1]	Description	Applicable Analysis Types[2]
MAXCPU TIME	Limit of CPU seconds	NL
MRESPONSE	Control printout of modal responses	FR
NLTYPE	Specify nonlinear static analysis type	NL
NODE ECHO	Echo node input	ALL1
ORTHOTROPIC	Orthotropic direction definition	SH, TH
RESEQUENCE	Element resequencing	ALL1
RESTART	Restart options	SH, TH
SAVE FILE	Save specific NISA files	ALL1, ALL2
SIGMA	Stefan-Boltzmann coefficient	SH, TH
SORT STRESS	Sort highest stresses	ST
STARTING TIME	Starting time	TR
STEP	Total number of steps for heat transfer analysis	TH
STRESS STIFFENING	Stress stiffening option	ST
WARNING	Set warning flag	ALL1, ALL2

(1) Minimum abbreviations are in bold face

(2) ST: Linear static
EV: Eigenvalue
BU: Buckling
NL: Nonlinear static
SH: Steady state heat transfer
TH: Transient heat transfer
TR: Transient dynamics
FR: Frequency response
RA: Random vibration
SHO: Shock spectrum
ALL1: All static and heat transfer analyses (ST, EV, BU, NL, SH, TH)
ALL2: All modal dynamic analyses (TR, FR, RA, SHO)

APPENDIX D

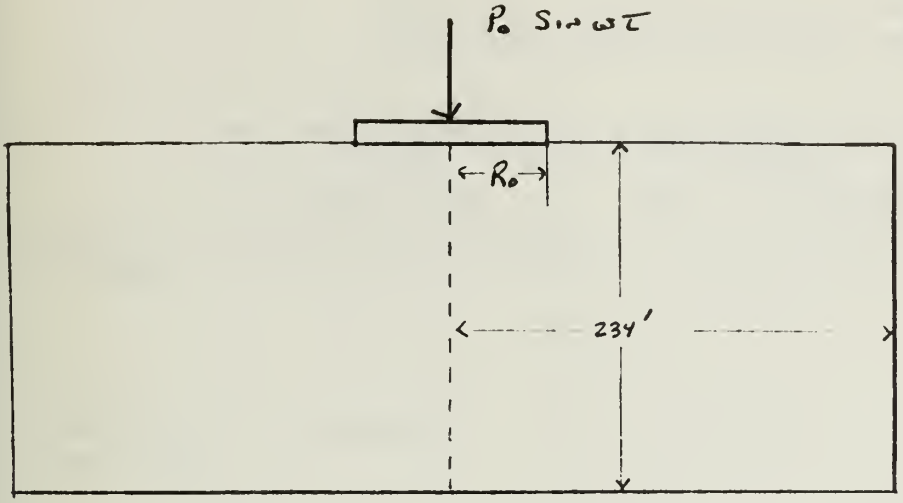
NISA II VERSES GTSTRUDL

DISPLACEMENT COMPARISONS

	VERTICAL DISPLACEMENT (INCHES)				HORIZONTAL DISPLACEMENT (INCHES)			
	NISA		GTSTRUDL		NISA		GTSTRUDL	
NODE	STATIC	DYNAMIC	STATIC	DYNAMIC	STATIC	DYNAMIC	STATIC	DYNAMIC
1	1.717×10^{-2}	2.611×10^{-7}			0	0		
2	1.689×10^{-2}	2.657×10^{-7}			0	0		
3	1.717×10^{-2}	1.521×10^{-7}			0	0		
4	1.694×10^{-2}	1.204×10^{-7}			0	0		
5	1.696×10^{-2}	1.807×10^{-7}			0	0		
6	1.694×10^{-2}	1.697×10^{-7}			0	0		
7	1.717×10^{-2}	2.444×10^{-7}			0	0		
8	1.689×10^{-2}	1.398×10^{-7}			0	0		
9	1.717×10^{-2}	1.215×10^{-7}			0	0		
22	1.807×10^{-2}	1.543×10^{-7}			0	3.371×10^{-3}		
23	2.756×10^{-2}	5.64×10^{-8}			0	3.372×10^{-3}		
24	1.807×10^{-2}	2.635×10^{-8}			0	3.713×10^{-3}		
25	1.807×10^{-2}	9.616×10^{-8}			0	3.371×10^{-3}		
26	2.756×10^{-2}	1.86×10^{-7}			0	3.372×10^{-3}		
27	1.802×10^{-2}	1.032×10^{-7}			0	3.371×10^{-3}		

APPENDIX E

THEORETICAL DISPLACEMENT CALCULATIONS



6 FT THICK CONCRETE CIRCULAR FOUNDATION ON SOIL

FDN MASS = 5145.4 $\#SEC^2/FT$ OR 0 $\#SEC^2/FT$

SOIL PROPERTIES

$\rho = 3.88$ $\#SEC^2/FT^4$

$G = 206{,}000$ psf

$\nu = 0.45$

FORCING FUNCTION $P_o \sin \omega t$

$P_o = 50{,}000$ #

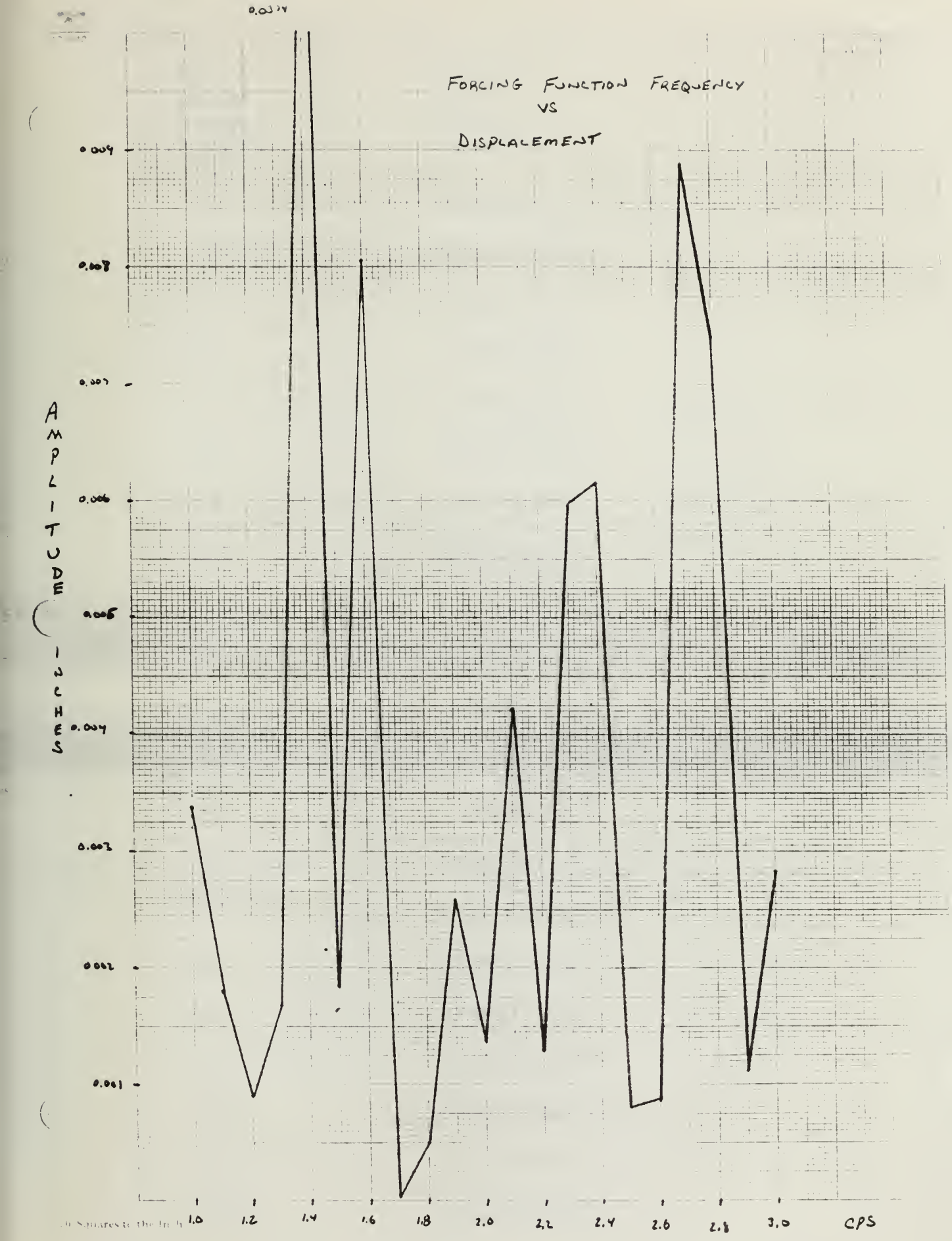
$\omega = 12.57$ RADIANS/SEC OR 18.85 RADIANS/SEC

CIRCULAR FOUNDATION ON A HALFSPACE
DISPLACEMENT CALCULATION SUMMARY

	f = 2 CPS FDN HAS MASS	f = 3 CPS FDN IS MASSLESS	f = 3 CPS FDN HAS MASS
NISA	0.00137 Ft	0.00231 Ft	0.00211 Ft
GAZETAS By Computer	0.00186 Ft	0.00174 Ft	0.00184 Ft
NAVFAC Hand Cal.	0.00344 Ft	0.00355 Ft (mass = 1×10^{-5} #sec²/Ft)	0.00332 Ft
LUCO Hand Cal		0.00344 Ft	
LYSMER Hand Cal	0.00916 Ft		0.00247 Ft

FOR 2CPS NISA II ANALYZED THE PROBLEM USING 33 EIGENVALUES STARTING WITH THE FUNDAMENTAL VALUE OF 0.635 CPS AND ENDING WITH 3.57 CPS. DUE TO THE LARGE DIFFERENCE BETWEEN LYSMER AND NISA, AND THE FACT THAT 2CPS IS ON THE PEAK OF LYSMER'S CURVE, A NISA ANALYSIS WAS PERFORMED ON THE MODEL WITH f = 1.0 TO 3.0 IN INCREMENTS OF 0.1 CPS. THE RESULTS ARE SHOWN ON THE FOLLOWING GRAPH

FOR 3CPS NISA II ANALYZED THE PROBLEM USING 40 EIGENVALUES RANGING FROM ABOUT 2.1 AND ENDING WITH ABOUT 4.2


FORCING FUNCTION FREQUENCY
VS
DISPLACEMENT
AMPLITUDE (INCHES)
0.009
0.008
0.007
0.006
0.005
0.004
0.003
0.002
0.001
1.0
1.2
1.4
1.6
1.8
2.0
2.2
2.4
2.6
2.8
3.0
CPS

Forcing Function Frequency (cycles/sec)	Foundation Center Displacement (inches)
1.0	3.338813×10^{-3}
1.1	1.801127×10^{-3}
1.2	9.005687×10^{-4}
1.3	1.672994×10^{-3}
1.4	3.742432×10^{-2}
1.5	1.840101×10^{-3}
1.6	8.047797×10^{-3}
1.7	3.945689×10^{-5}
1.8	5.197638×10^{-4}
1.9	2.562883×10^{-3}
2.0	1.373975×10^{-3}
2.1	4.226132×10^{-3}
2.2	1.285588×10^{-3}
2.3	5.962294×10^{-3}
2.4	6.135445×10^{-3}
2.5	8.096274×10^{-4}
2.6	8.681937×10^{-4}
2.7	8.828307×10^{-3}
2.8	7.418144×10^{-3}
2.9	1.142477×10^{-3}
3.0	2.810870×10^{-3}

EIGENVALUES USED FOR THE 2CPS NISA ANALYSIS

NUMBER	RAD/SEC	CYC/SEC	RATIO
******	*******	*******	*****
1	3.990103E+00	6.350446E-01	0.000000E-01
2	5.141340E+00	8.182696E-01	0.000000E-01
3	6.500898E+00	1.034650E+00	0.000000E-01
4	6.730453E+00	1.071185E+00	0.000000E-01
5	8.702491E+00	1.385045E+00	0.000000E-01
6	8.804094E+00	1.401215E+00	0.000000E-01
7	1.017533E+01	1.619454E+00	0.000000E-01
8	1.128642E+01	1.796290E+00	0.000000E-01
9	1.137774E+01	1.810823E+00	0.000000E-01
10	1.183149E+01	1.883040E+00	0.000000E-01
11	1.348654E+01	2.146450E+00	0.000000E-01
12	1.372117E+01	2.183792E+00	0.000000E-01
13	1.384859E+01	2.204072E+00	0.000000E-01
14	1.448125E+01	2.304763E+00	0.000000E-01
15	1.522606E+01	2.423303E+00	0.000000E-01
16	1.545846E+01	2.460290E+00	0.000000E-01
17	1.590396E+01	2.531194E+00	0.000000E-01
18	1.617381E+01	2.574141E+00	0.000000E-01
19	1.697778E+01	2.702097E+00	0.000000E-01
20	1.744101E+01	2.775824E+00	0.000000E-01
21	1.847239E+01	2.939973E+00	0.000000E-01
22	1.874823E+01	2.983873E+00	0.000000E-01
23	1.899211E+01	3.022689E+00	0.000000E-01
24	1.971806E+01	3.138227E+00	0.000000E-01
25	1.977497E+01	3.147284E+00	0.000000E-01
26	2.018043E+01	3.211815E+00	0.000000E-01
27	2.038338E+01	3.244116E+00	0.000000E-01
28	2.100639E+01	3.343270E+00	0.000000E-01
29	2.143819E+01	3.411994E+00	0.000000E-01
30	2.167573E+01	3.449800E+00	0.000000E-01
31	2.212480E+01	3.521272E+00	0.000000E-01
32	2.225399E+01	3.541833E+00	0.000000E-01
33	2.241003E+01	3.566667E+00	0.000000E-01

$f = 2$ cps

NAVFAC / RICHART METHOD w/ GAZETAS K FOR HAS MASS

$$K_v = \frac{4GR}{1-\nu}\left(1 + 1.28\,\frac{R}{H}\right)$$

$$= \frac{(4)(206000\ psf)(9\ ft)}{(1-0.45)}\left[1 + 1.28\left(\frac{9}{234}\right)\right]$$

$$= 14{,}147{,}446.2\ \#/FT$$

$G = 206000$ psf
$R = 9'$
$H = 234'$
$\nu = 0.45$

$$f_n = \frac{1}{2\pi}\sqrt{\frac{K_v}{m}}$$

$$= \frac{1}{2\pi}\sqrt{\frac{14147446.2\ \#/FT}{5145.4\ \#\,Sec^2/FT}}$$

$$= 8.35\ \text{cycles/sec}$$

W FOU = 165682 # $\therefore m = 5145.4\ \frac{\#\,Sec^2}{FT}$

MASS RATIO

$$B_o = \left(\frac{1-\nu}{4}\right)\left(\frac{m}{\rho R^3}\right)$$

$$= \left(\frac{(1-0.45)}{4}\right)\left(\frac{5145.4\ \#\,Sec^2/FT}{(3.88\ \#\,Sec^2/ft^4)(9\ ft)^3}\right)$$

$$= 0.25$$

$m = 5145.4\ \#\,Sec^2/Ft$
$\rho = 3.88\ \#\,Sec^2/ft^4$

SHEAR WAVE VELOCITY

$$V_s = \sqrt{\frac{G}{\rho}} = \sqrt{\frac{206000\ \#/ft^2}{3.88\ \#\,Sec^2/ft^4}} = 230.4\ ft/sec$$

$$f_s = \frac{V_s}{4H} = \frac{230.4\ ft/sec}{4(234\ ft)} = 0.246\ \text{cycles/sec}$$

$f = 2$ cycles/sec $> f_s$

ASSUME V_s ROCK $\neq \infty$ & $\frac{V_s}{V_R} \simeq 0.5$

$f = 2$ cps $\therefore$ $f/f_s = 8.12$ $\therefore$ Rigid layer has little to no effect + System Damping $\approx$ Half Space Damping

$\frac{C_v}{C_{HS}} = 1$

$\therefore$ Damping Ratio $D_z = \frac{0.425}{\sqrt{B_e}}$

$= \frac{0.425}{\sqrt{0.25}} = 0.85$

Magnification Factor

$$M = \left[\left(1-\left(\frac{f}{f_n}\right)^2\right)^2 + \left(2(D_z)\left(\frac{f}{f_n}\right)\right)^2\right]^{-1/2}$$

$$= \sqrt{\frac{1}{\left(1-\left(\frac{2}{8.35}\right)^2\right)^2 + \left(2(0.85)\left(\frac{2}{8.35}\right)\right)^2}}$$

$= 0.974$

$$A_s = \frac{P_o}{K_v} = \frac{50{,}000\ \#}{14147\ \#/ft} = 0.00353\ Ft$$

$$\delta_{Dyn} = A_s \cdot M = (0.00353\ Ft)(0.974) = 0.00344\ Ft$$

```
**********************************************
*                                            *
*            DYNAMIC ANALYSIS                *
*         using ' GAZETAS ' METHOD           *
*    Earthquake engineering vol.2 no1,1983   *
*                                            *
*                                            *
*            IBM version 1.0                 *
*                                            *
*                                            *
**********************************************
```

f = 2 cps FDN HAS MASS

```
*******************************************
*        I n p u t  -  d a t a            *
*******************************************
```

MASS DENSITY OF SOIL (KIPS-SEC^2/FT^4)= .00388

DEPTH TO BEDROCK [H] (FT) = 234

FOUNDATION EMBEDMENT [D] (FT) = 0

SHEAR MODULUS OF SOIL (KSF) = 206

Data for little [k] and little [c] :

HORIZONTAL DYNAMIC STIFFNESS [KH] = 1

VERTICAL DYNAMIC STIFFNESS [KV] = .97

ROCKING DYNAMIC STIFFNESS [KR] = .94

TORSIONAL DYNAMIC STIFFNESS [KT] = .96

HORIZONTAL-ROCKING STIFFNESS [KHR] = 1

VERTICAL DAMPING COEFFICIENT [CV] = .84

HORIZONTAL DAMPING COEFFICIENT [CH] = .59

ROCKING DAMPING COEFFICIENT [CR] = .11

HORIZONTAL-ROCKING DAMPING COEFFICIENT [CHR] = 0

TORSIONAL DAMPING COEFFICIENT [CT] = 9.000001E-02

POISSON RATIO = .45

ROCKING MASS OF INERTIA - CG /AXIS X (KIP-SEC^2-FT) = 119.631

TORSION MASS OF INERTIA - CG /AXIS Z (KIP-SEC^2-FT) = 208.388

MASS OF FOUNDATION + MACHINERY (KIP-SEC^2/FT) = 5.145

TORSIONAL EXCITING MOMENT (FT-KIP) = 0

HORIZONTAL EXCITING FORCE (KIPS) = 0

ROCKING EXCITING MOMENT (FT-KIP) = 0

VERTICAL EXCITING FORCE (KIPS) = 50

EQUIVALENT RADIUS OF FOUNDATION (FT) = 9

OPERATING FREQUENCY OF SYSTEM (RPM) = 120

VERTICAL PHASE ANGLE IN DEGREES = 0

HORIZONTAL PHASE ANGLE IN DEGREES = 0

ROCKING PHASE ANGLE IN DEGREES = 0

TORSIONAL PHASE ANGLE IN DEGREES = 0

HYSTERETIC MATERIAL DAMPING (%)= 0

DISTANCE FROM CENTER OF GRAVITY TO BASE OF FOUNDATION (FT)= 3

****
* OUTPUT - DATA *
****

HORIZONTAL STATIC STIFFNESS (KIP/FT)= 9753.052

VERTICAL STATIC STIFFNESS (KIP/FT)= 26172.78

HORIZONTAL-ROCKING STATIC STIFFNESS (KIP/FT)= 0

TORSIONAL STATIC STIFFNESS (KIP/FT)= 800928.1

ROCKING STATIC STIFFNESS (KIP/FT)= 732783.8

VERTICAL DISPLACEMENT (FT)= 1.862894E-03 ←

VERTICAL PHASE ANGLE (RAD)= -.4137561

TORSIONAL DISPLACEMENT (FT)= 0

TORSIONAL PHASE ANGLE (RAD)= -4.803612E-02

HORIZONTAL DISPLACEMENT (FT)= 0

HORIZONTAL PHASE ANGLE (RAD)= 1.20483

ROTATIONAL DISPLACEMENT (FT)= 0

ROTATIONAL PHASE ANGLE (RAD)= 1.20483

***** End of Dynamic Analysis *****

```
R(1) =  9
R(3) =  9
R(5) =  9
R(7) =  9
R(9) =  9
AI(1) =  .2895918
AI(3) =  .4123002
AI(5) =  5.399169E-02
AI(7) =  0
AI(9) =  4.417502E-02
KI(1) =  9753.052
KI(2) =  2824.404
KI(3) =  25387.6
KI(4) =  10791.04
KI(5) =  688816.8
KI(6) =  39564.24
KI(7) =  0
KI(8) =  0
KI(9) =  768891
KI(10) =  35381.02
HR= 8940.587  HI= 2824.404  CR=-29259.16  CI=-8473.211  RR= 757702.9  RI=
 64983.87
NR= 5.806465E+09            NI= 2.225215E+09            V1= 24575.13  V2=
 10791.04     V3= 26839.96  V4= .4137561  V5= 1.862894E-03
PV=-.4137561  O1= 735983.7  O2= 35381.02  O3= 736833.6  O4= 4.803612E-02
    O5= 0     PO=-4.803612E-02
RA= 760484.4  RB= 8.555496E-02            CA= 30461.35  CB= 3.423471  HA=
 9376.106     HB= .3059867
H1= 0         H2= 8.555496E-02            H3= 0         H4= 3.423471  H5= 0
    H6= 0
H7= 0         H8= 0         H9= 0         PH= 1.20483
I1= 0         I2= 0         I3= 0         I4= 1.570796  I5= 6.218248E+09
    I6= .3659669
R1= 0         R2= .3059867  R3= 0         R4= 3.423471  R5= 0         R6= 0
R7= 0         R8= 0         R9= 0
S1= 0         S2= 0         S3= 0         S4= 1.570796
PR= 1.20483
End of internal storage data
```

LYSMER FOUNDATION HAS MASS

$f = 2$ CPS

$a_0 = \omega \frac{r_0}{V_s}$ $\quad r_0 = 9'$ $\quad \omega = (2\text{ cps})(2\pi \text{ rad/c}) = 12.57$ rad/sec

$V_s = \sqrt{\frac{G}{\rho}} = \sqrt{\frac{206000 \text{ psf}}{288 \frac{\text{slugs}}{\text{ft}^3}}} = 230$ ft/sec

$= (12.57)\left(\frac{9 \text{ ft}}{230 \text{ ft/sec}}\right)$

$= 0.49$

$\therefore$ FROM CHART $M = 2.47$

$$\delta_{MAX} = \frac{(1-\mu) P_0}{4G r_0} M = \frac{(1-0.45)(50{,}000\#)}{4(206000 \text{ psf})(9\text{ ft})} 2.47$$

$= 0.00916$ Ft

$f = 3$ CPS

$\therefore a_0 = (3\text{cps})(2\pi \text{ rad/c})\left(\frac{9\text{ ft}}{230 \text{ ft/sec}}\right) = 0.74$

$\therefore$ FROM CHART $M = 0.74$

$$\delta_{MAX} = \frac{(1-\mu) P_0}{4G r_0} M = \frac{(1-0.45)(50{,}000\#)}{4(206000 \text{ psf})(9\text{ ft})}(0.74)$$

$= 0.00274$ Ft

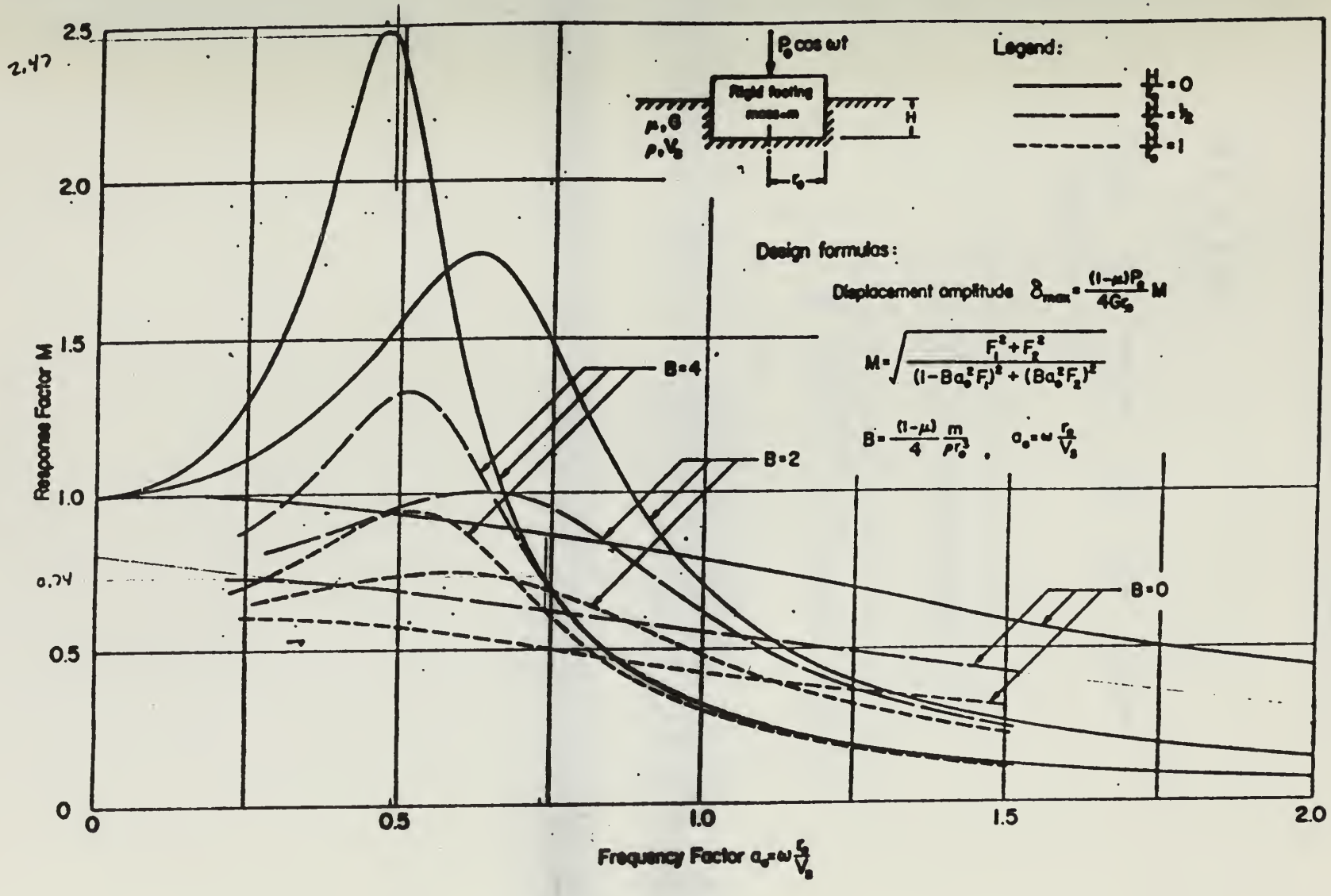


FIG. 17 – STEADY STATE SPECTRA FOR EMBEDDED FOOTINGS

NAVFAC / RICHART METHOD W/ GAZETAS K

$f = 3$ cps

SAME AS NAVFAC FOR 2 cps EXCEPT DIFFERENT m

$$\therefore \quad m = \frac{1}{\left[\left(1-\left(\frac{f}{f_n}\right)^2\right)^2 + \left((2)(D_s)\left(\frac{f}{f_n}\right)\right)^2\right]^{1/2}}$$

$$= \frac{1}{\left[\left(1-\left(\frac{3}{8.35}\right)^2\right)^2 + \left((2)(0.85)\left(\frac{3}{8.35}\right)\right)^2\right]^{1/2}}$$

$$= 0.94$$

$$\delta = A_s\, m = \frac{P_o}{K_i}\, m = \left(\frac{50{,}000}{14{,}973{,}962.2}\right)(0.94) = 0.00332 \text{ ft}$$

```
**************************************************
*                                                *
*            DYNAMIC ANALYSIS                    *
*         using ' GAZETAS ' METHOD               *
*    Earthquake engineering vol.2 no1.1983       *
*                                                *
*                                                *
*            IBM version 1.0                     *
*                                                *
*                                                *
**************************************************
```

f = 3 cps FDN HAS MASS

```
*********************************************
*        I n p u t - d a t a                *
*********************************************
```

MASS DENSITY OF SOIL (KIPS-SEC^2/FT^4)= .00388

DEPTH TO BEDROCK [H] (FT) = 234

FOUNDATION EMBEDMENT [D] (FT) = 0

SHEAR MODULUS OF SOIL (KSF) = 206

Data for little [k] and little [c] :

HORIZONTAL DYNAMIC STIFFNESS [KH] = .99

VERTICAL DYNAMIC STIFFNESS [KV] = .9

ROCKING DYNAMIC STIFFNESS [KR] = .9

TORSIONAL DYNAMIC STIFFNESS [KT] = .92

HORIZONTAL-ROCKING STIFFNESS [KHR] = 1

VERTICAL DAMPING COEFFICIENT [CV] = .85

HORIZONTAL DAMPING COEFFICIENT [CH] = .6

ROCKING DAMPING COEFFICIENT [CR] = .1

HORIZONTAL-ROCKING DAMPING COEFFICIENT [CHR] = 0

TORSIONAL DAMPING COEFFICIENT [CT] = 9.000001E-02

POISSON RATIO = .45

TORSION MASS OF INERTIA - CG /AXIS Z (KIP-SEC^2-FT) = 208.388

MASS OF FOUNDATION + MACHINERY (KIP-SEC^2/FT) = 5.145

TORSIONAL EXCITING MOMENT (FT-KIP) = 0

HORIZONTAL EXCITING FORCE (KIPS) = 0

ROCKING EXCITING MOMENT (FT-KIP) = 0

VERTICAL EXCITING FORCE (KIPS) = 50

EQUIVALENT RADIUS OF FOUNDATION (FT) = 9

OPERATING FREQUENCY OF SYSTEM (RPM) = 180

VERTICAL PHASE ANGLE IN DEGREES = 0

HORIZONTAL PHASE ANGLE IN DEGREES = 0

ROCKING PHASE ANGLE IN DEGREES = 0

TORSIONAL PHASE ANGLE IN DEGREES = 0

HYSTERETIC MATERIAL DAMPING (%)= 0

DISTANCE FROM CENTER OF GRAVITY TO BASE OF FOUNDATION (FT)= 3

****
* OUTPUT - DATA *
****

HORIZONTAL STATIC STIFFNESS (KIP/FT)= 9753.052

VERTICAL STATIC STIFFNESS (KIP/FT)= 26172.78

HORIZONTAL-ROCKING STATIC STIFFNESS (KIP/FT)= 0

TORSIONAL STATIC STIFFNESS (KIP/FT)= 800928.1

ROCKING STATIC STIFFNESS (KIP/FT)= 732783.8

VERTICAL DISPLACEMENT (FT)= 1.837587E-03

VERTICAL PHASE ANGLE (RAD)= -.6459611

TORSIONAL DISPLACEMENT (FT)= 0

TORSIONAL PHASE ANGLE (RAD)= -7.989973E-02

HORIZONTAL DISPLACEMENT (FT)= 0

```
ROTATIONAL PHASE ANGLE  (RAD)=  .9749001

***** End of Dynamic Analysis *****

VS= 230.4187
R(1) =  9
R(3) =  9
R(5) =  9
R(7) =  9
R(9) =  9
AI(1) =  .4417502
AI(3) =  .6258128
AI(5) =  7.362503E-02
AI(7) =  0
AI(9) =  6.626253E-02
KI(1) =  9655.521
KI(2) =  4308.412
KI(3) =  23555.5
KI(4) =  16379.26
KI(5) =  659505.4
KI(6) =  53951.23
KI(7) =  0
KI(8) =  0
KI(9) =  736853.8
KI(10) =  53071.52
HR= 7827.477  HI= 4308.412  CR=-28966.57  CI=-12925.24  RR= 703899.6  RI=
 92726.94
NR= 4.436252E+09            NI= 3.009706E+09            V1= 21727.45  V2=
 16379.26     V3= 27209.6    V4= .6459611  V5= 1.837587E-03
PV=-.6459611  01= 662812.5   02= 53071.52  03= 664933.8  04= 7.969973E-02
    05= 0     PO=-7.989973E-02
RA= 709980.9  RB= .130979    CA= 31719.45  CB= 3.56129   HA= 8934.864  HB=
 .5031669
H1= 0         H2= .130979    H3= 0         H4= 3.56129   H5= 0         H6= 0
H7= 0         H8= 0          H9= 0         PH= .9749001
I1= 0         I2= 0          I3= 0         I4= 1.570796  I5= 5.362501E+09
    I6= .5958963
R1= 0         R2= .5031669   R3= 0         R4= 3.56129   R5= 0         R6= 0
R7= 0         R8= 0          R9= 0
S1= 0         S2= 0          S3= 0         S4= 1.570796
PR= .9749001
End of internal storage data
```

NAVFAC / RICHART METHOD W/ GAZETAS K f = 3CPS

ASSUME FDN HAS 0 MASS ∴ SET FDN M = 1×10^{-5}

FROM PREVIOUS CALC. $K_V = 14147446.2$ #/FT

$$f = \frac{1}{2\pi}\sqrt{\frac{K}{m}} = \frac{1}{2\pi}\sqrt{\frac{14147446.2}{1\times10^{-5}}} = 189304 \text{ CPS}$$

MASS RATIO

$$B_z = \left(\frac{1-\nu}{4}\right)\left(\frac{m}{\rho R_0^3}\right) = \left(\frac{1-0.45}{4}\right)\left(\frac{1\times10^{-5}}{(3.98)(9)^3}\right) = 4.86\times10^{-10}$$

SHEAR WAVE = 230.4 FT/SEC (FROM BEFORE)

$f_s = 0.246$ C/S (FROM BEFORE)

$f = 3\text{CPS} > f_s$ ∴ ASSUME V_S ROCK ≠ 00 & $\frac{V_S}{V_R} = 0.5$

$\frac{f}{f_n} = 12.2$ ∴ RIGID LAYER HAS NO EFFECT

∴ $\frac{C_y}{C_{y0}} = 1$

∴ DAMPING RATIO = $\frac{0.425}{\sqrt{B_z}} = \frac{0.425}{\sqrt{4.86\times10^{-10}}} = 19276.0$

$$m = \frac{1}{\left[\left(1-\left(\frac{f}{f_n}\right)^2\right)^2 + \left(2(D_z)\left(\frac{f}{f_n}\right)\right)^2\right]} = \frac{1}{\left[\left(1-\left(\frac{3}{189304}\right)^2\right)^2 + \left((2)(19276)\left(\frac{3}{189304}\right)\right)^2\right]^{1/2}}$$

$= 0.999 = 1$

$$\delta = A_3 m = \frac{P_0}{K} m = \left(\frac{50{,}000}{14147446.2}\right)(1) = 0.00353 \text{ FT}$$

LUCO

MASSLESS FOUNDATION

$f = 3$ CPS $\quad V = 50{,}000$ lbs $\quad \mu = G = 206000$ psf

$a_0 = \omega a (\rho/\mu)^{1/2} \quad a = 9$ Ft $\quad \rho = 3.88 \frac{\#\,Sec^2}{Ft^4}$

$\Delta V = C_{VV}(a_0)V \quad \nu = \tau = 0.45$

$(3\ C/S)(2\pi\ Rad/c) = 18.85$ RADIANS/SEC

$$a_0 = (18.85\ Rad/Sec)(9\ Ft)\left(\frac{3.88\ \#Sec^2/Ft^4}{206000\ \#/Ft^2}\right)^{1/2} = 0.74$$

$\therefore$ FROM LUCO CHART

$$\frac{4}{1-\tau}\mu a\, C_{VV}(a_0)_{INS.} = 0.76 = X_I$$

$$\frac{4}{1-\tau}\mu a\, C_{VV}(a_0)_{REM} = 0.53 = X_R$$

$$X = \frac{4}{1-\tau}\mu a\, C_{VV}\, a_0$$

$$\therefore C_{VV} = \frac{X}{\left(\frac{4}{1-\tau}\right)(\mu)(a)(a_0)}$$

$$\therefore C_{VV\ INS} = \frac{0.76}{\left(\frac{4}{1-0.45}\right)(206000\ psf)(9\ Ft)(0.74)} = 7.617 \times 10^{-8}\ Ft/\#$$

$$\therefore C_{VV\ REM} = \frac{0.53}{\left(\frac{4}{1-0.45}\right)(206000\ psf)(9\ Ft)(0.74)} = 5.312 \times 10^{-8}\ Ft/\#$$

LUCO MASSLESS FDN CONT.

$$\Delta V_i = C_{VV_i}\, q_0\, V$$

$$= (7.617 \times 10^{-8}\ Ft/\#)(0.74)(50000\ \#)$$

$$= 0.00282\ Ft$$

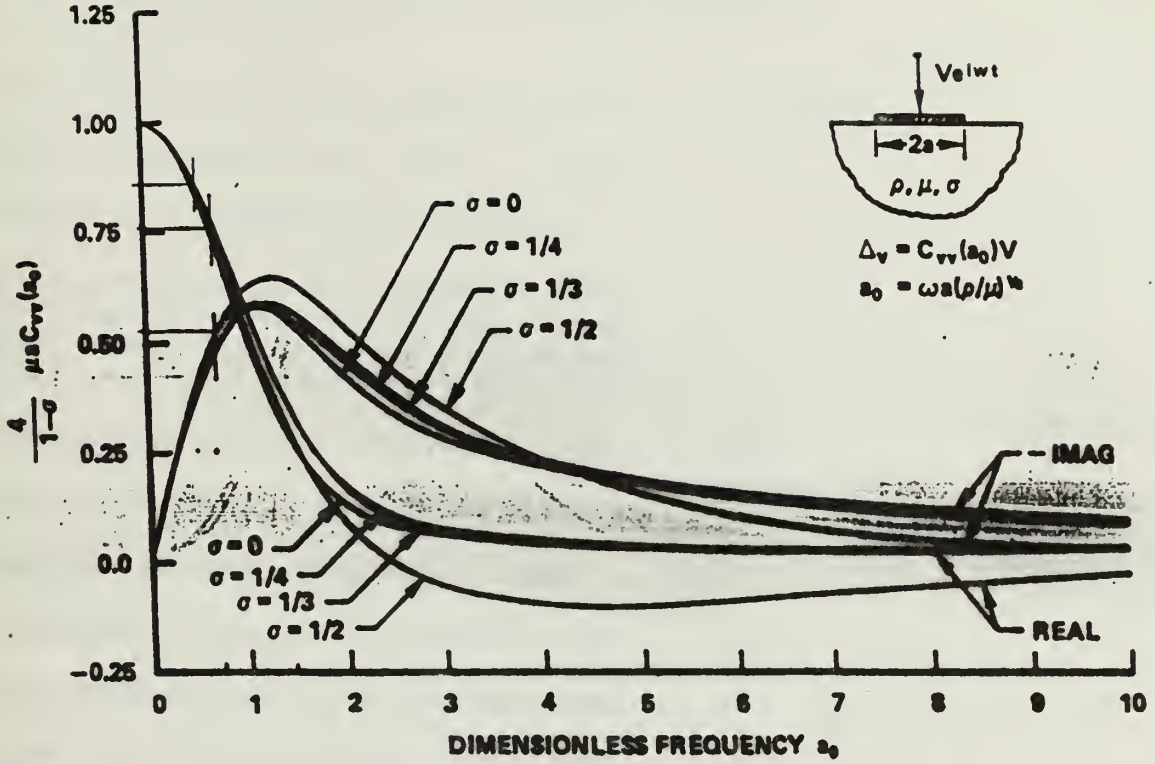
$$\Delta V_R = C_{VV_R}\, q_0\, V$$

$$= (5.312 \times 10^{-8}\ Ft/\#)(0.74)(50000\ \#)$$

$$= 0.00197\ Ft$$

$$\Delta V_{Total} = \sqrt{(V_i)^2 + (V_R)^2}$$

$$= 0.00344\ Ft$$


1.25
1.00
0.75
0.50
0.25
0.0
-0.25
$\frac{4}{1-\sigma}\ \mu a C_{vv}(a_0)$
0
1
2
3
4
5
6
7
8
9
10
DIMENSIONLESS FREQUENCY a_0
$\sigma = 0$
$\sigma = 1/4$
$\sigma = 1/3$
$\sigma = 1/2$
IMAG
REAL
Velwt
2a
ρ, μ, σ
$\Delta_v = C_{vv}(a_0)V$
$a_0 = \omega a(\rho/\mu)^{1/2}$

DYNAMIC ANALYSIS
using ' GAZETAS ' METHOD
Earthquake engineering vol.2 no1,1983

IBM version 1.0

f = 3cps FDN HAS NO MASS

Input - data

MASS DENSITY OF SOIL (KIPS-SEC^2/FT^4)= .00388

DEPTH TO BEDROCK [H] (FT) = 234

FOUNDATION EMBEDMENT [D] (FT) = 0

SHEAR MODULUS OF SOIL (KSF) = 206

Data for little [k] and little [c] :

HORIZONTAL DYNAMIC STIFFNESS [KH] = .99

VERTICAL DYNAMIC STIFFNESS [KV] = .9

ROCKING DYNAMIC STIFFNESS [KR] = .9

TORSIONAL DYNAMIC STIFFNESS [KT] = .92

HORIZONTAL-ROCKING STIFFNESS [KHR] = 1

VERTICAL DAMPING COEFFICIENT [CV] = .85

HORIZONTAL DAMPING COEFFICIENT [CH] = .6

ROCKING DAMPING COEFFICIENT [CR] = .1

HORIZONTAL-ROCKING DAMPING COEFFICIENT [CHR] = 0

TORSIONAL DAMPING COEFFICIENT [CT] = 9.000001E-02

POISSON RATIO = .45

MASS OF FOUNDATION + MACHINERY (KIP-SEC^2/FT) = 0

TORSIONAL EXCITING MOMENT (FT-KIP) = 0

HORIZONTAL EXCITING FORCE (KIPS) = 0

ROCKING EXCITING MOMENT (FT-KIP) = 0

VERTICAL EXCITING FORCE (KIPS) = 50

EQUIVALENT RADIUS OF FOUNDATION (FT) = 9

OPERATING FREQUENCY OF SYSTEM (RPM) = 180

VERTICAL PHASE ANGLE IN DEGREES = 0

HORIZONTAL PHASE ANGLE IN DEGREES = 0

ROCKING PHASE ANGLE IN DEGREES = 0

TORSIONAL PHASE ANGLE IN DEGREES = 0

HYSTERETIC MATERIAL DAMPING (%)= 0

DISTANCE FROM CENTER OF GRAVITY TO BASE OF FOUNDATION (FT)= 3

****
* OUTPUT - DATA *
****

HORIZONTAL STATIC STIFFNESS (KIP/FT)= 9753.052

VERTICAL STATIC STIFFNESS (KIP/FT)= 26172.78

HORIZONTAL-ROCKING STATIC STIFFNESS (KIP/FT)= 0

TORSIONAL STATIC STIFFNESS (KIP/FT)= 800928.1

ROCKING STATIC STIFFNESS (KIP/FT)= 732783.8

VERTICAL DISPLACEMENT (FT)= 1.742741E-03

VERTICAL PHASE ANGLE (RAD)= -.6075967

TORSIONAL DISPLACEMENT (FT)= 0

TORSIONAL PHASE ANGLE (RAD)= -7.190033E-02

HORIZONTAL DISPLACEMENT (FT)= 0

```
ROTATIONAL PHASE ANGLE  (RAD)=  1.069473

***** End of Dynamic Analysis *****

VS= 230.4187
R(1) =  9
R(3) =  9
R(5) =  9
R(7) =  9
R(9) =  9
AI(1) =  .4417502
AI(3) =  .6258128
AI(5) =  7.362503E-02
AI(7) =  0
AI(9) =  6.626253E-02
KI(1) =  9655.521
KI(2) =  4308.412
KI(3) =  23555.5
KI(4) =  16379.26
KI(5) =  659505.4
KI(6) =  53951.23
KI(7) =  0
KI(8) =  0
KI(9) =  736853.8
KI(10) =  53071.52
HR= 9655.521  HI= 4308.412  CR=-28966.57  CI=-12925.24  RR= 746405.1  RI=
 92726.94
NR= 6.135424E+09          NI= 3.362348E+09          V1= 23555.5   V2=
 16379.26     V3= 28690.44  V4= .6075967  V5= 1.742741E-03
PV=-.6075967  O1= 736853.8  O2= 53071.52  O3= 738762.6  O4= 7.190033E-02
    O5= 0     PO=-7.190033E-02
RA= 752142.8  RB= .1235981  CA= 31719.45  CB= 3.56129   HA= 10573.15  HB=
 .4196996
H1= 0         H2= .1235981  H3= 0         H4= 3.56129   H5= 0         H6= 0
H7= 0         H8= 0         H9= 0         PH= 1.069473
I1= 0         I2= 0         I3= 0         I4= 1.570796  I5= 6.996343E+09
    I6= .5013234
R1= 0         R2= .4196996  R3= 0         R4= 3.56129   R5= 0         R6= 0
R7= 0         R8= 0         R9= 0
S1= 0         S2= 0         S3= 0         S4= 1.570796
PR= 1.069473
End of internal storage data
```

LUDLEY KNOX LIBRARY
NAVAL POSTGRADUATE SCHOOL
MONTEREY CA 93943-5002

Thesis
C622 Collins
c.1 Utilization of a PC for
finite element modeling.

Thesis
C622 Collins
c.1 Utilization of a PC for
finite element modeling.

AMERICAN BINDERIES 3

thesC622
Utilization of a PC for finite element m
3 2768 000 83299 2
DUDLEY KNOX LIBRARY